CALIFORNIA INSTITUTE OF TECHNOLOGY

EARTHQUAKE ENGINEERING RESEARCH LABORATORY

FRAME3D V2.0 – A PROGRAM FOR THE THREE-DIMENSIONAL NONLINEAR TIME-HISTORY ANALYSIS OF STEEL STRUCTURES: USER GUIDE

BY

SWAMINATHAN KRISHNAN

REPORT NO. EERL 2009-04

PASADENA, CALIFORNIA

MAY 2009

A REPORT ON RESEARCH SUPPORTED BY THE CALIFORNIA INSTITUTE OF TECHNOLOGY UNDER THE SUPERVISION OF SWAMINATHAN KRISHNAN

Abstract

This is Version 2.0 of the user guide and should be used along with Version 2.0 of the program. Updates include: 1. Realistic PMM interaction surfaces for plastic hinge elements (output file PMM). 2. 5-Segment modified elastofiber element for brace and slender column modeling. 3. Eigen value problem solver using subspace iteration (output files MODES and EIGEN). 4. Output the sum of forces of groups of elements (output file ELMGRPRES). Additional input is required as a result of these additions to the program. However, the example input files shown in chapter 6 correspond to the input format from Version 1.0 and do not reflect the changes in the input file from Version 1.0 to Version 2.0.

Updates in Version 1.1 include: 1. Output files FRAC, FRACSUM, and FRACTOT, summarizing the fractures in the elastofiber beam elements; 2. Output file RUP listing the fibers that have ruptured during the course of the analysis; 3. Output file FAIL listing the elastofiber elements that have a complete segment failure; 4. Output files FEMA356 and PERF summarizing the performance of the beams, columns, and panel zones, relative to the Federal Emergency Management Agency document FEMA356 (FEMA 2000) acceptance criteria; 5. Output files XDRFT, YDRFT, AVGPKDRFT, and PKDRFT listing the average and peak interstory drifts in the building. Additional input is required for this output processing. In addition, some typographical errors in the version 1.0 of the user guide were also corrected, the most notable of these being sections 4.1.8 and 4.1.9 dealing with elastofiber element fiber fracture.

FRAME3D is a program for the three-dimensional nonlinear analysis of steel buildings. It aims to overcome the computational challenges posed by full 3D analysis of steel buildings subject to earthquake ground motion through efficient finite elements that are designed to capture the essence of material behavior and geometry evolution. The element library consists of a plastic hinge beam element, an elastofiber beam element, a 5-segment modified elastofiber element, a panel zone element, a 4-noded diaphragm element to model floor slabs, and an elastic translational/rotational spring element to model foundations and compliant supports. The program utilizes a Newton-Raphson iteration strategy applied to an implicit Newmark time-integration scheme to solve the nonlinear equations of motion at each time-step. Geometric nonlinearity and shear deformation are included in the formulation. This document serves as a User Guide to the program. All the input and output variables encountered by the user are described here along with brief descriptions of the various types of elements. In addition, 2 examples illustrating the capabilities and usage of the program are presented. Finally a glossary of all the variables is alphabetically listed at the end of the document.

Contents

List of Figures

List of Tables

Chapter 1 Introduction

FRAME3D is a comprehensive 3D building analysis program ideally suited for analyzing steel-framed buildings subject to earthquake motions. A variety of elements suitable for nonlinear modeling of structural elements in a building are provided. A brief outline of the program with its capabilities is given in Chapter 2. The element library with a brief description of each element is given in Chapter 3. The format for the user input files is provided in Chapter 4 along with the description of all input variables. Program output is described in Chapter 5. Two examples illustrating the usage and capabilities of the program are provided in Chapter 7. Finally, an alphabetically ordered glossary of input and output variables is provided for the user's convenience in Appendix A.

For detailed information on the implementation of the program, the user is referred to Krishnan (2003), Krishnan and Hall (2006a, 2006b), and Krishnan (2009).

This program has been successfully used to simulate collapse in 20-story buildings subject to strong ground motion from earthquakes on desktop computers. It is believed that it can be used for the analysis of 40-60 story buildings with similar computing resources.

Chapter 2 Program Outline

The following are the key features and capabilities of the program:

1. Stepped nonlinear static 3D analysis of buildings.

2. Nonlinear dynamic 3D time-history analysis of buildings.

3. Eigen analysis using the subspace iteration algorithm (Bathe 1996) to identify a subset of the model natural frequencies and corresponding mode shapes.

4. Element library includes inelastic beam elements, inelastic panel zone elements, elastic diaphragm elements to model floors and elastic spring elements to recover reactions.

5. P-Delta effects are automatically included (through geometry updating). This also enables large deformation problems to be solved.

6. Shear deformation is included in beam element formulation.

7. Effect of axial force on moment capacity of beam elements is included.

8. Stiffness matrix is stored in banded form thus minimizing storage requirements.

9. Damping matrix is of the proportional Rayleigh damping type. It could be mass proportional or stiffness proportional or a combination of the two.

10. Option to iterate with the initial elastic stiffness matrix or with the tangent stiffness matrix or a combination of both.

11. Time-histories of nodal displacements, element forces and inelastic deformation can be written out and can be plotted using standard plotting programs.

12. Element peak forces and inelastic deformations can be written out and maps of plastic deformation of various frames can be created using a program such as matlab to gain an understanding of the damage observed in the structure during severe earthquakes.

13. The deformed shape of the structure in the form of updated nodal coordinates can be written out at any desired time interval. Using a program such as matlab, an animation of the structure's shaking during an earthquake can then be generated.

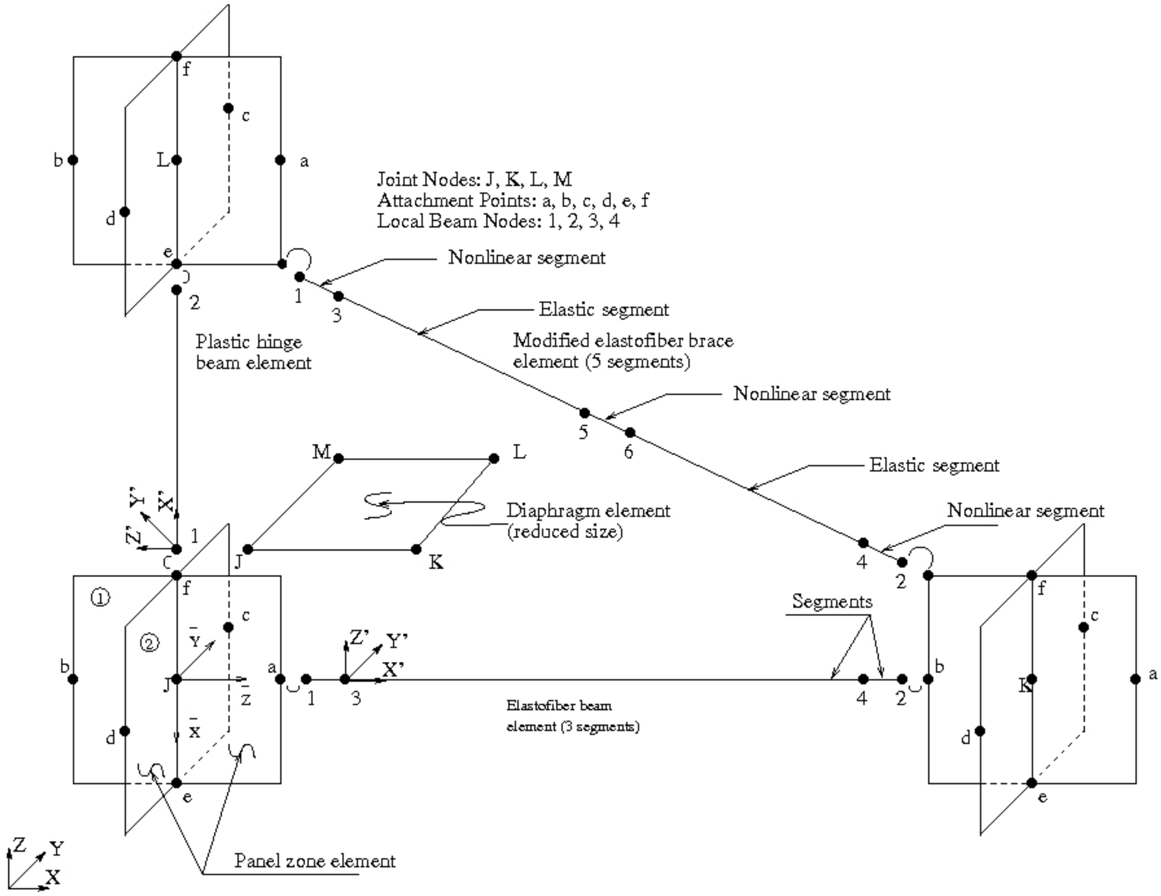


Figure 2.1: Element Arrangement in Frame Model, Showing Nodes, Attachment Points, and Coordinate Systems

2.1 Description of Model

The program can be utilized for the modeling of 3D frame structures consisting of grids of beams and columns. The structural model is comprised of three element types: panel zone elements, beam elements for beams and columns, and braces, and diaphragm elements for floor slabs (Figure 2.1).

2.1.1 Panel Zone Element

This element models nonlinear shear deformation in the region of the joint where the beams and columns intersect. The joint region consists of a length of column within the depth of the connecting beams. The shear deformation is due primarily to opposing moments from the beams and columns at the joint caused by the frame being subjected to lateral loads (Figure 2.2). Each panel zone element is associated with a global node J, K, etc. at the center of the joint where the global degrees of freedom (DOF) are defined. Each panel zone element consists of two orthogonal panels ① and ② which always remain planar and orthogonal. Edges of these panels contain attachment points a, b, c, and d, where beams attach, and e and f on the top and bottom where columns attach. Braces attach to the corners of the panels. Details of the panel zone

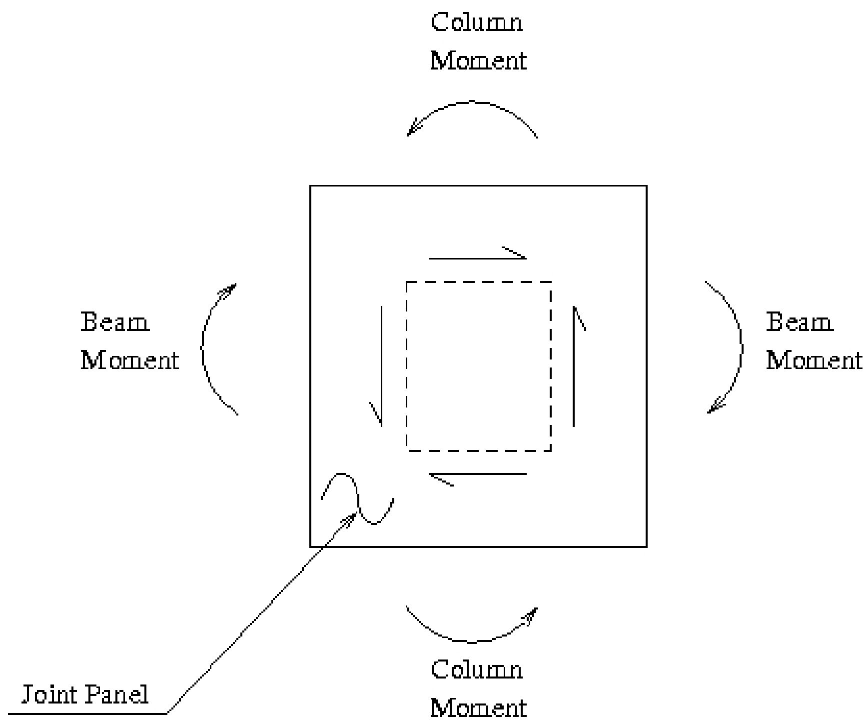


Figure 2.2: Panel zone shear caused by beam and column moments at the joint

element are presented in Chapter 3.

2.1.2 Beam Element

This element is used to model beams, columns, and braces. Three types of beam elements have been developed: plastic hinge type, elastofiber type, and modified elastofiber (MEF) type. The plastic hinge element has 2 nodes with local node numbers 1 and 2, while the elastofiber element consists of 3 segments with 4 nodes. The exterior nodes at the ends of the element are numbered 1 and 2, while the interior nodes are numbered 3 and 4. Both elements consider nonlinear behavior for flexural and axial deformations. The MEF element is a 5-segment version of the elastofiber element. It consists of three fiber segments, two at the member ends and one at mid-span, with two elastic segments sandwiched in between. The fiber segments are divided into 20 fibers in the cross-section that run the length of the segment. The fibers exhibit nonlinear axial stress-strain behavior. Member-end yielding and mid-span buckling, with fracture or rupture of fibers leading to complete severing of the brace, can be accurately modeled. Thus, the element is suitable for modeling slender columns and braces that are sensitive to buckling. This element can also be used to model beams with significant axial loads. Details of the plastic hinge, elastofiber, and MEF elements are given in Chapter 3.

2.1.3 Diaphragm Element

This element is used to model the in-plane stiffness of floor slabs. It is essentially a 4-noded plane-stress element connecting to global nodes J, K, L, and M. Details of the diaphragm element are presented in Chapter 3.

2.1.4 Coordinate Systems

The following are the various coordinate systems used in the structural model:

1. XYZ: The global coordinate system, XYZ, is the fixed coordinate system that is used to define the structure in space. The coordinates of the nodes defined by the user are in this coordinate system. In general it is recommended that Z axis be taken to be vertical (direction of gravity) and the X and Y axes be oriented in the two principal directions of the building.

2. $\bar{X}\bar{Y}\bar{Z}$: This coordinate system is local to the panel zone element. $\bar{X}$ lies along the line connecting attachment points e and f, in the direction of a vector running from f to e. $\bar{X} - \bar{Z}$ is the plane of panel ① and $\bar{X} - \bar{Y}$ is the plane of panel ②. This coordinate system translates and rotates with the panel zone element.

3. $X'Y'Z'$: This coordinate system is local to the plastic hinge beam element. X' lies along the longitudinal axis of the beam running from node 1 to node 2. $Y' - Z'$ is the cross-sectional plane of the beam. Y' is the major principal axis of the cross-section while Z' is the minor axis. This coordinate system translates and rotates with the beam element.

4. $X'Y'Z'$: This coordinate system is local to each segment of the elastofiber beam element. Its orientation is similar to the plastic hinge beam element except that each segment has its own $X'Y'Z'$ system. Each translates and rotates with the corresponding segment of the element.

5. $X'Y'$: This coordinate system is local to the plane of the diaphragm element with nodes J, K, L, and M. Its origin lies at the center of the element. X' axis is in the direction of the vector running from node K to L and Y' is perpendicular to X' and in the plane of the element.

2.1.5 Structure Degrees of Freedom

The global degrees of freedom (DOF) are associated only with the nodes J, K, etc. at the centers of joints. The structural model consists of 8 global DOF at each node, J. These DOF are listed below:

1. U_{JX}: Translation in the X direction.

2. U_{JY}: Translation in the Y direction.

3. U_{JZ}: Translation in the Z direction.

4. $\theta_{J\bar{X}}$: Rotation of panel zone element as a rigid body about the $\bar{X}$ axis.

5. $\theta^{B}_{J\bar{Y}}$: Rotation of the line $e - f$ about the $\bar{Y}$ axis. It corresponds to deformation of panel ① into the shape of a parallelogram (Figure 2.3a). The B indicates that the rotating panel edges are connected to beams.

6. $\theta^{C}_{J\bar{Y}}$: Rotation of the line $a - b$ about the $\bar{Y}$ axis. It corresponds to deformation of panel ① into the shape of a parallelogram (Figure 2.3b). The C indicates that the rotating panel edges are connected to columns. Note that $\theta^{B}_{J\bar{Y}}$ and $\theta^{C}_{J\bar{Y}}$ together accommodate a rigid rotation of panel ① about $\bar{Y}$ plus its shear deformation.

7. $\theta^{B}_{J\bar{Z}}$: Rotation of the line $e - f$ about the $\bar{Z}$ axis. It corresponds to deformation of panel ② into the shape of a parallelogram (Figure 2.3c). The B indicates that the rotating panel edges are connected to beams.

8. $\theta^{C}_{J\bar{Z}}$: Rotation of the line $c - d$ about the $\bar{Z}$ axis. It corresponds to deformation of panel ② into the shape of a parallelogram (Figure 2.3d). The C indicates that the rotating panel edges are connected to columns. Note that $\theta^{B}_{J\bar{Z}}$ and $\theta^{C}_{J\bar{Z}}$ together accommodate a rigid rotation of panel ② about $\bar{Z}$ plus its shear deformation.

Panel zone elements contribute stiffness directly to the four global DOF, $\theta^{B}_{J\bar{Y}}$, $\theta^{C}_{J\bar{Y}}$, $\theta^{B}_{J\bar{Z}}$, and $\theta^{C}_{J\bar{Z}}$. Beam elements are formulated in terms of 3 translational and 3 rotational DOF at each of the local nodes (1 and 2 for plastic hinge element and 1, 2, 3, and 4 for elastofiber element. However, none of these DOF appear in the global equations. Appropriate stiffness terms are assembled into the global DOF through transformation matrices based on the geometry of the panel zone elements at nodes 1 and 2.

In addition to material nonlinearity, geometric stiffness (effect of axial force on flexural stiffness) is included with the beam elements. In addition, as the building deforms, locations of global nodes, local beam nodes and attachment points are updated, as are all the local coordinate systems. This ensures that $P - \Delta$ effects are accounted for accurately.

Mass is lumped into the global translational DOF at nodes J, K, etc. Damping is assumed to be of the Rayleigh type (stiffness and mass proportional). Building foundation is assumed to be rigid.

2.2 Equations of Motion

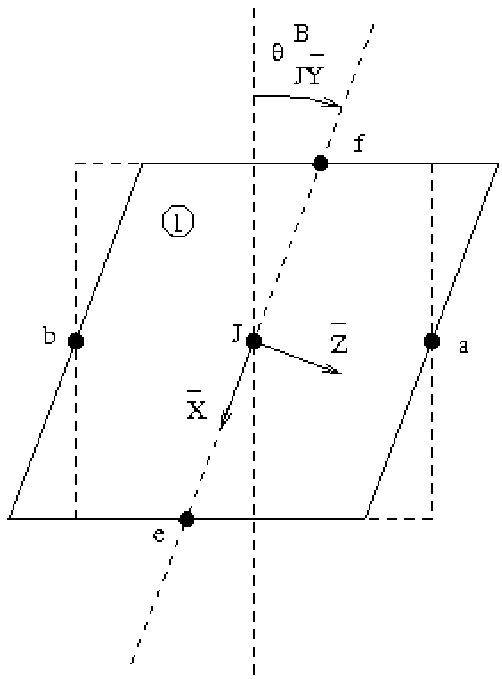
The matrix equation of motion of the building (Cook et al. 1989) as a function of time, t, is:

$$[M]\left\{\ddot{U}(t)\right\} + [C]\left\{\dot{U}(t)\right\} + \{R(t)\} = \{f_g\} - [M]\,[r]\left\{\ddot{U}_g(t)\right\} \tag{2.1}$$

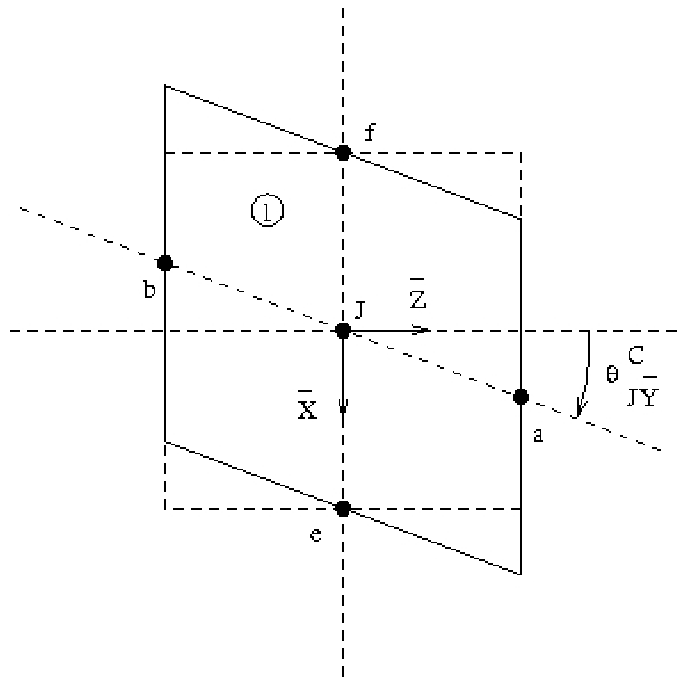
In the above,

1. $\{U(t)\}$ = Vector of global displacements at time, t, comprising of the set of 8 global translations and rotations at each node, J, K, etc.

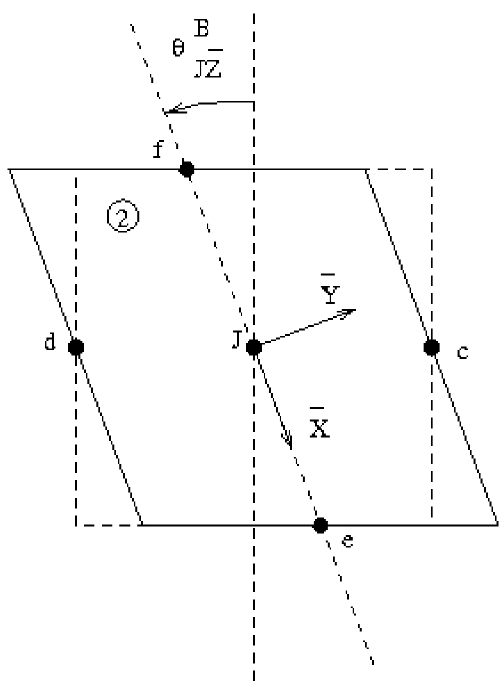
2. $\left\{\dot{U}(t)\right\}$, $\left\{\ddot{U}(t)\right\}$ = Vectors of nodal velocities and accelerations respectively corresponding to the 8 global translations and rotations at each node.



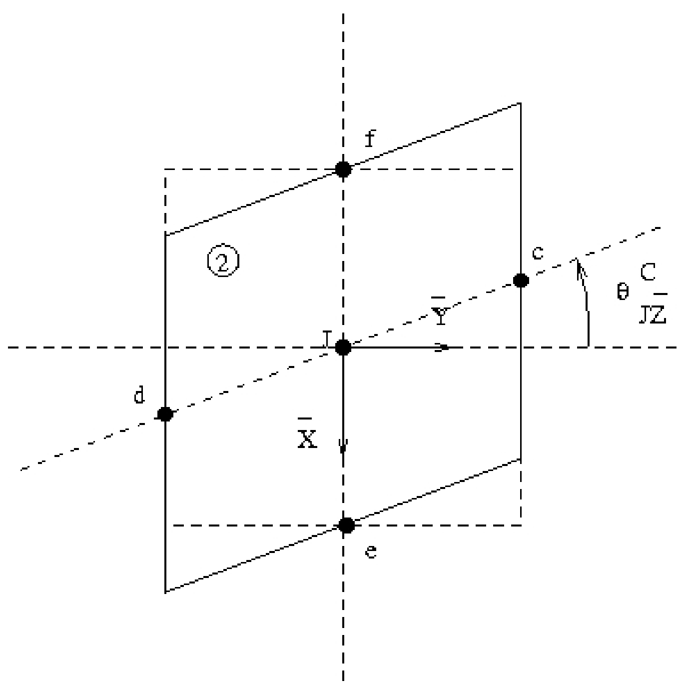
(a) $\theta^{B}_{J\bar{Y}}$ DOF



(b) $\theta^{C}_{J\bar{Y}}$ DOF



(c) $\theta^{B}_{J\bar{Z}}$ DOF



(d) $\theta^{C}_{J\bar{Z}}$ DOF

Figure 2.3: Degrees of Freedom of Panel Zone Element

3. $[M]$ = Structure mass matrix. The mass in the structural system is lumped into the translational degrees of freedom at the joint nodes rendering the mass matrix diagonal. Further, the rotary inertia of the lumped nodal masses is neglected, and so the only non-zero terms of the mass matrix are the diagonal terms corresponding to the translational degrees of freedom.

4. $[C]$ = Structure damping matrix.

   Damping is assumed to be of the Rayleigh type (stiffness and mass proportional). Thus, the damping matrix is computed as

$$[C] = \alpha_0 [M] + \alpha_1 [K] \tag{2.2}$$

   where a_0 and a_1 are user defined proportionality constants. The initial elastic stiffness matrix is used in the above computation.

   In the linear elastic case, where modal superposition is valid, if Rayleigh damping is assumed, the modal damping in the i^{th} mode can be written in terms of the corresponding modal frequency as

$$\zeta_i = \frac{1}{2}\left(\alpha_0 \omega_i + \frac{\alpha_1}{\omega_i}\right) \tag{2.3}$$

   If we plot ω_i vs ζ_i, the curve is found to be reasonably flat in the intermediate frequency range. Assuming that the structure exhibits approximately equal damping values in all the modes, two frequencies, ω and $R\omega$, are picked covering the frequency range of interest and a desired amount of damping. Then a_0 and a_1 can be computed as

$$\alpha_0 = \frac{2\zeta\omega R}{1+R} \tag{2.4a}$$

$$\alpha_1 = \frac{2\zeta}{\omega(1+R)} \tag{2.4b}$$

5. $\{R(t)\}$ = Vector of stiffness forces corresponding to the configuration, $\{U(t)\}$. It is computed considering all material and geometric nonlinear effects.

6. $\{f_g\}$ = Vector of static gravity loads for which a static analysis is performed first.

7. $\left\{\ddot{U}_g(t)\right\}$ = Vector consisting of two horizontal components (X and Y) and one vertical component (Z) of the free-field ground acceleration at time t. The ground motion is assumed uniform.

8. $[r]$ = A 3-column matrix of zeroes except for ones in the first, second, and third columns in the positions corresponding to the X, Y, and Z DOF, respectively.

9. The base DOF are fixed and are not included.

Chapter 3 Element Library

The element library of program, FRAME3D, is listed in Table 3.1.

Table 3.1: FRAME3D Element Library

S.No.	Element
1	Panel zone element.
2	Plastic-hinge beam element.
3	Elastofiber beam element.
4	Modified elastofiber beam element.
5	Elastic diaphragm element.
6	Elastic translational/rotational spring element.

3.1 Panel Zone Element

Each joint is modeled by a panel zone element consisting of a length of column within the depth of the connecting beams. This implies that one column, the associated column of the joint (ACJ) runs continuously through the height of the joint. The panel zone element is an idealization of the joint region of this column. It consists of two rectangular panels which are perpendicular to each other, panel ① in the $\bar{X}-\bar{Z}$ plane and panel ② in the $\bar{X}-\bar{Y}$ plane, forming a cruciform section (Figure 3.1). The thicknesses of all web plates and web doubler plates of the ACJ are combined to form the thickness of panel ①, $t_p^{①}$, while the thicknesses of all flange plates of the ACJ are combined to form the thickness of panel ②, $t_p^{②}$. The depth, D, of the joint is taken to be equal to the depth (dimension along the minor axis, Z') of the ACJ. The width, W, of the joint is taken to be equal to the width (dimension along the major axis, Y') of the ACJ. The height, H, of the joint is taken to be equal to the depth of the deepest beam framing into that joint. Beams and columns modeled using beam elements connect to the panel zone element at the mid-points of the edges of the two panels. These connection points are henceforth referred to as attachment points. There are six attachment points, a through f. Attachment points a, b, c, and d are reserved for beams, while attachment points e and f are reserved for columns. Braces are connected to the panel corners.

The panels are assumed to deform only in shear as a result of the end moments and shears of the attached beams and columns. However, they remain planar and perpendicular to each other at all times. The relation between the shear stress, τ, and the shear strain, γ, can be a nonlinear relation. Detailed experimental (Kato 1982; Krawinkler and Popov 1982; Popov and Petersson 1978; Bertero et al. 1972) and analytical (Pinkney 1973; Krawinkler et al. 1975; Kato 1982; Popov and Petersson 1978; Fielding and Chen 1973) studies have been carried out on steel-frame joints and have resulted in a much better understanding of their hysteretic behavior. Two material shear stress-strain models (bilinear and linear-quadratic (Challa 1992; Hall and Challa 1995)) are implemented in this work.

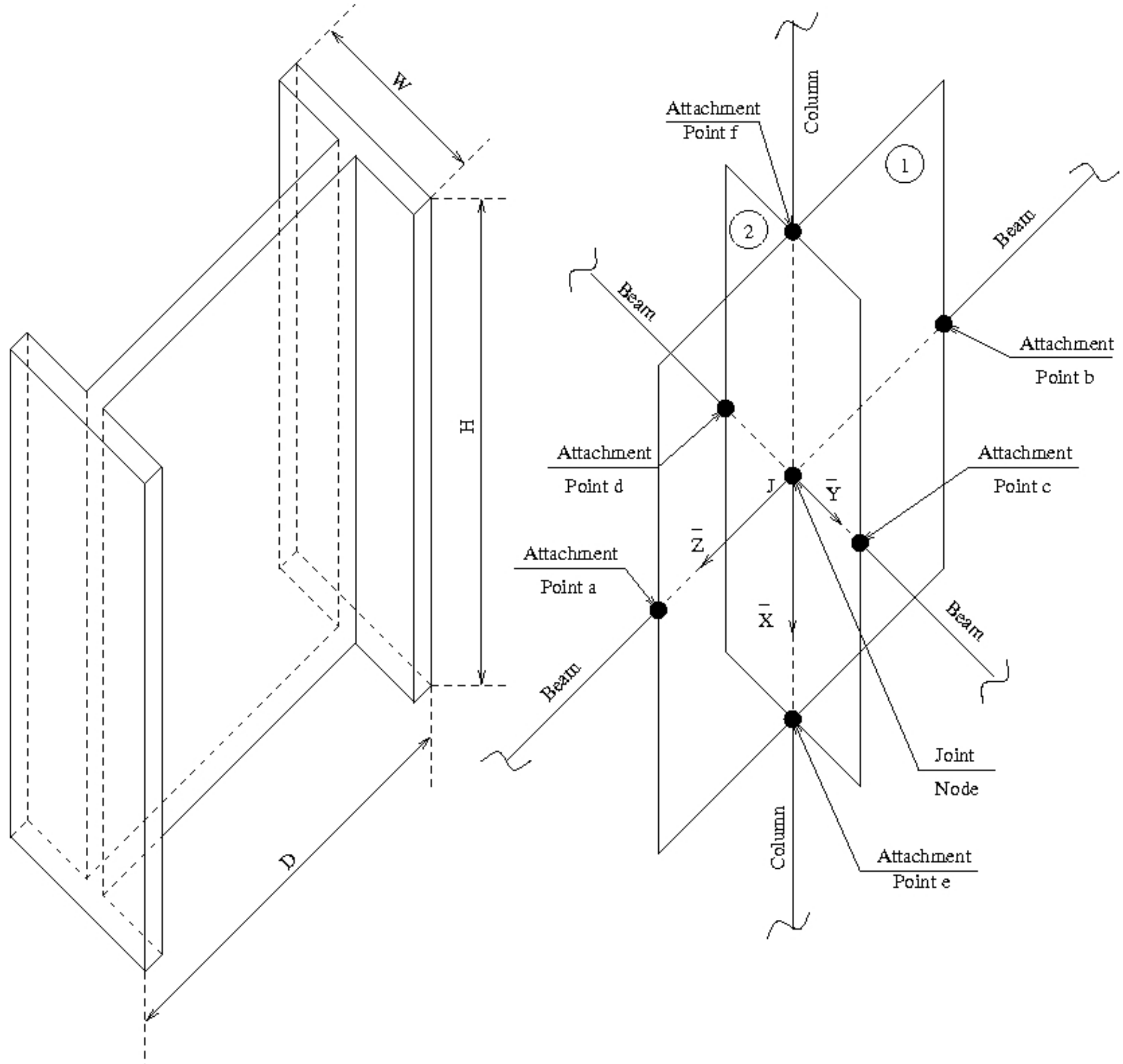


Figure 3.1: Panel Zone Element

Each panel has two degrees of freedom as shown in Figure 2.3: $\theta^B_{J\bar{Y}}$ and $\theta^C_{J\bar{Y}}$ for panel ①, and $\theta^B_{J\bar{Z}}$ and $\theta^C_{J\bar{Z}}$ for panel ②, where J is the global node at the center of the joint. These DOF are also global DOF. Strain or rotation in one of the panels causes a rigid body rotation but no strain in the orthogonal panel.

The joint coordinate system with origin at the center of the joint (at the node), $\bar{X}\bar{Y}\bar{Z}$ is defined as follows:

1. $\bar{X}$ axis: Defined as the vector from attachment point f to attachment point e of the joint.

2. $\bar{Y}$ axis: It is a vector normal to the plane consisting of attachment points a, b, e and f. A unit vector in the direction of $\bar{Y}$ axis is computed as a cross-product of the vector going from point b to point a with the vector going from point f to point e and dividing by the length of the resulting vector.

3. $\bar{Z}$ axis: It is a vector normal to the plane consisting of attachment points c, d, e, and f. A unit vector in the direction of $\bar{Y}$ axis is computed as a cross-product of the vector going from point d to point c with the vector going from point f to point e and dividing by the length of the resulting vector.

Thus, at the beginning of the analysis, the joint coordinate system is the same as the local element coordinate system of the associated column of the joint (ACJ).

3.1.1 Material Models for Panel Shear

Bilinear Model

The bilinear model was first proposed by Fielding and Chen (1973). The virgin shear stress-strain curve for the joint hysteresis behavior in this model is bilinear (Figure 3.2). The virgin curves for positive and negative shears are identical. The shear yield stress, τ_y, is given by the von Mises yield criterion:

$$\tau_y = \frac{\sigma_y}{\sqrt{3}} \tag{3.1}$$

where σ_y is the axial yield stress of the material. The yield strain is given by

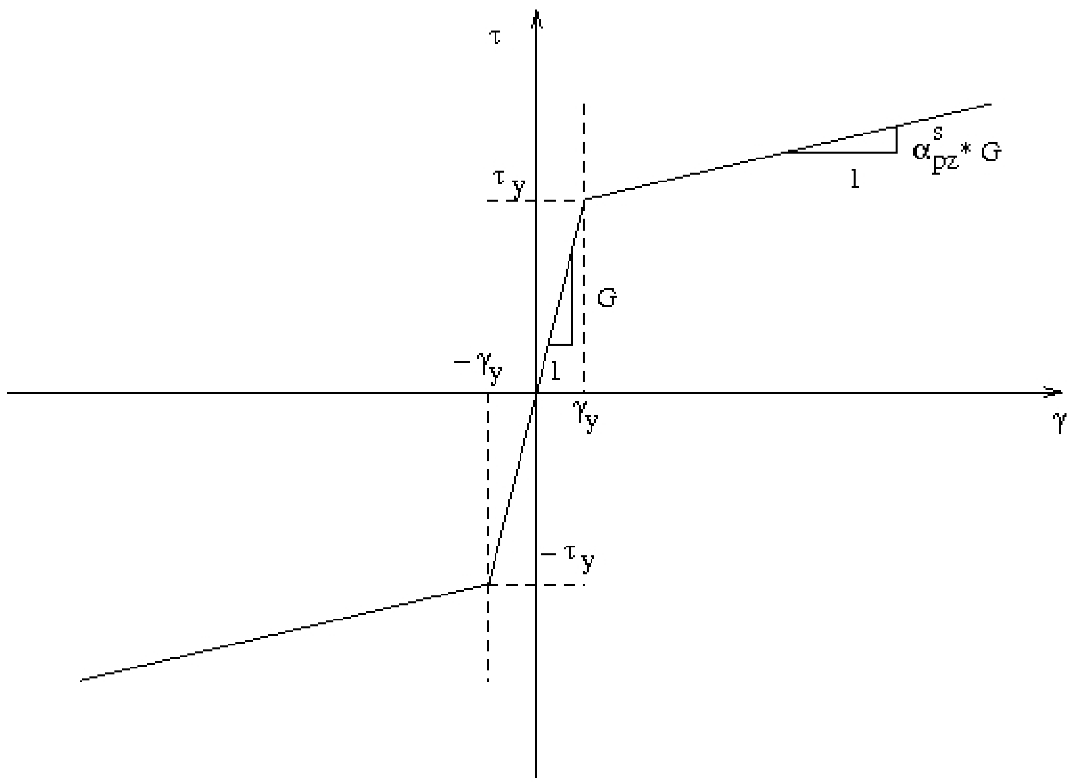
$$\gamma_y = \frac{\tau_y}{G} \qquad where \qquad G = \frac{E}{2(1+\nu)} \tag{3.2}$$

In the above, E is the Young's modulus, G is the shear modulus, and ν is the Poisson's ratio of the material. For a panel zone with the bilinear material model, the tangent shear modulus is given by:

$$G_T = G \text{ if panel zone has not yielded} \tag{3.3a}$$

$$G_T = \alpha^s_{pz} G \text{ if panel zone has yielded} \tag{3.3b}$$

The ratio, α^s_{pz}, of the post-yield stiffness to the shear modulus modulus, is user-defined. Unloading occurs linearly until zero stress is reached at a rate equal to the elastic modulus and further loading in the reverse direction follows a bilinear stress-strain relation in the opposite direction shifted suitably.



Bilinear shear stress–strain law for a panel zone

Figure 3.2: Shear Stress-Strain Model for a Bilinear Panel Zone Element

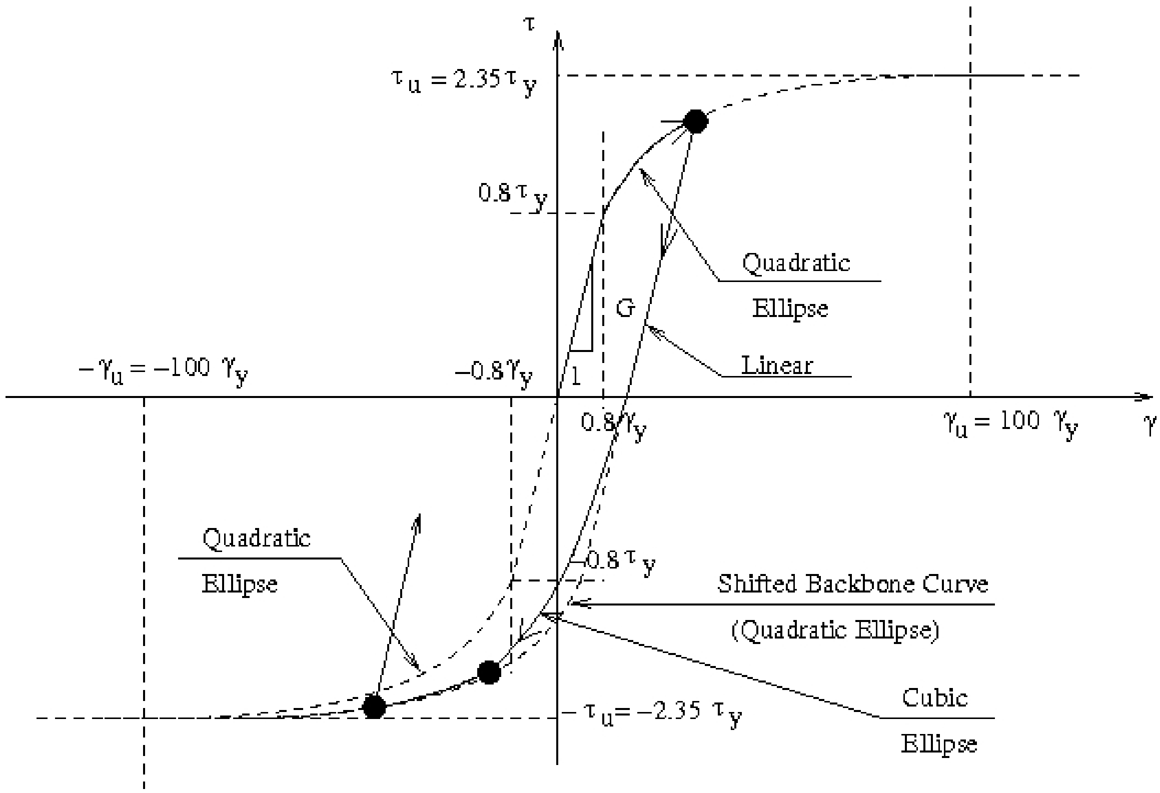


Figure 3.3: Shear Stress-Strain Model for a Linear-Quadratic Panel Zone Element

Linear-Quadratic Ellipsoidal Model

The linear-quadratic ellipsoidal joint hysteresis model proposed by Hall and Challa (1992, 1995) defines a virgin curve for the joint hysteresis behavior as shown in Figure 3.3. The shear stress-strain is assumed to be linear until a stress of $0.8\tau_y$ is reached corresponding to a strain of $0.8\gamma_y$, where, as before, τ_y and γ_y are the yield shear stress and strain respectively. The ultimate shear stress, τ_u, of the panel zone is assumed to be equal to $2.35\tau_y$, corresponding to an ultimate shear strain, γ_u, of $100\gamma_y$. The post-ultimate behavior is assumed to be perfectly plastic. The curve between the joint shear strain limits of $0.8\gamma_y$ and $100\gamma_y$ is defined by a quadratic ellipse given by:

$$\frac{(\gamma - \gamma_0)^2}{a^2} + \frac{(\tau - \tau_0)^2}{b^2} = 1 \tag{3.4}$$

where σ_0 and γ_0 are the coordinates of the center of the ellipse whose semi-major and semi-minor axes are a and b, respectively. The hysteresis loops or the branch curves are defined by cubic ellipses given by:

$$-\frac{(\gamma - \gamma_0)^3}{a^3} + \frac{(\tau - \tau_0)^3}{b^3} = 1 \text{ for positive curve} \tag{3.5a}$$

$$\text{and } \frac{(\gamma - \gamma_0)^3}{a^3} - \frac{(\tau - \tau_0)^3}{b^3} = 1 \text{ for negative curve} \tag{3.5b}$$

The hysteresis rules to define the cyclic response of each panel are given in Challa (1992). For a panel zone with the linear-quadratic ellipsoidal material model, the tangent shear modulus, G_T, is given by the slope of the curve (Figure 3.3) at the current state which is determined from the stress-strain history and current strain increment.

3.2 Plastic Hinge Beam Element

The following assumptions are made in the formulation of this element:

1. The cross-section is assumed uniform along the length of the element.
2. The cross-section is assumed to be doubly symmetric (with shear center at the centroid).
3. Plane sections remain plane; however, they do not have to remain normal to the beam axis. Thus, shear deformations are included.
4. Strains in the element are small.
5. Lateral deflections relative to the chord are small. This means that the effect of bowing (Oran 1973a; Oran 1973b; Kassimali 1983) on axial stiffness is neglected. This effect is important only for very slender members or during post-buckling of compression members such as braces in braced frames.
6. Warping restraint under twisting is neglected.
7. The element is loaded only at its ends.
8. Composite action with the slab is not included.
9. Local buckling is not included.

The plastic hinge beam element has two nodes which connect to the attachment points a through f of the panel zone element. This element can model beams and columns in framed structures. Columns connect to attachment points e and f while beams connect to attachment points a through d. The beam element along with its degrees of freedom is shown in Figure3.4. Original length of the element is L_0.

The plastic hinge element local coordinate system, $X'Y'Z'$, is a right-handed, orthogonal system and is defined as follows:

1. X' axis: It runs along the longitudinal axis of the element at the centroid of the cross-section. It is defined as a vector from node 1 to node 2 which are also located at the centroid.
2. Y' axis: Y' axis is orthogonal to X' and is the major principal axis of the cross-section of the element.
3. Z' axis: Z' axis is the minor principal axis of the cross-section of the element.

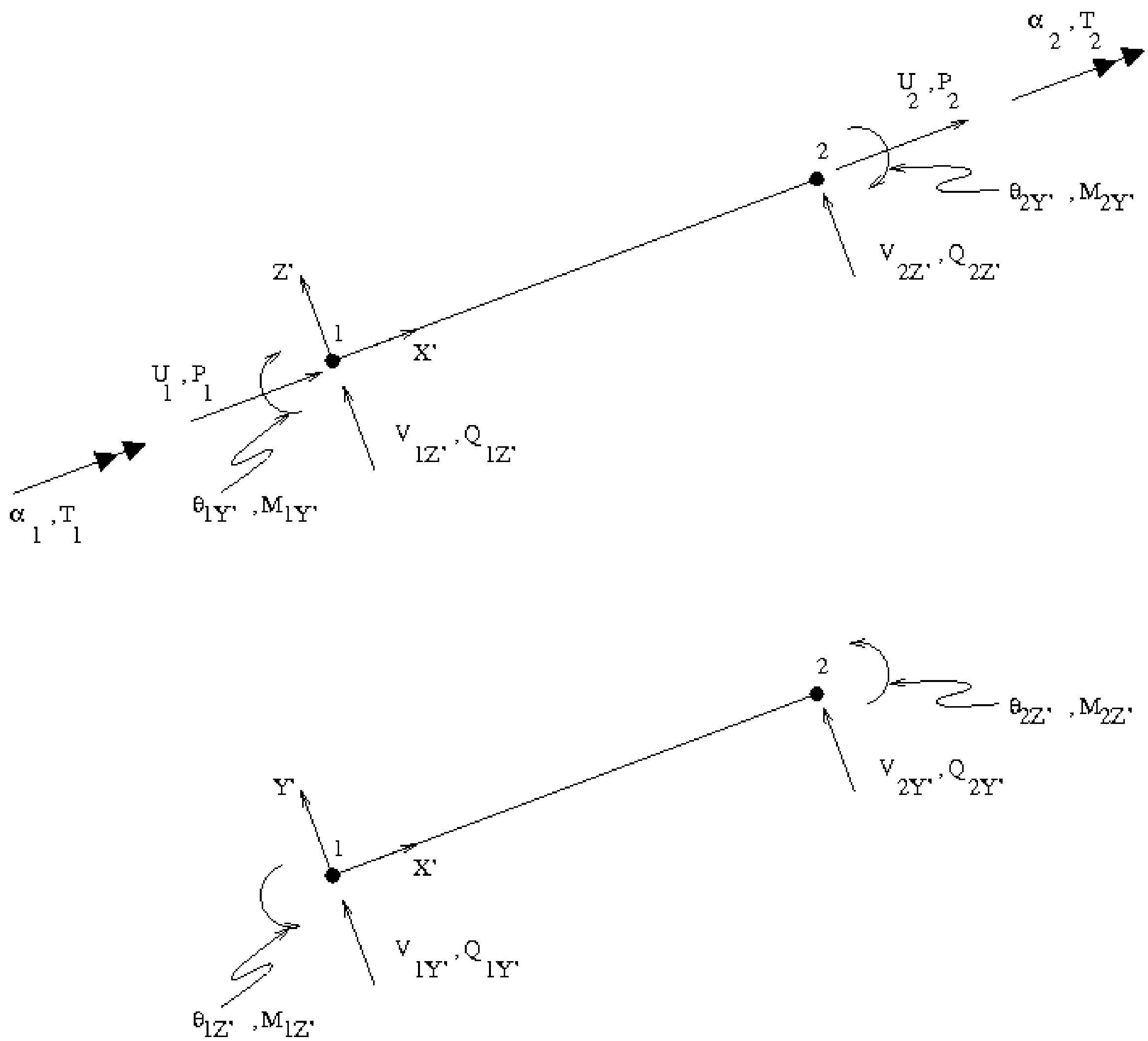


Figure 3.4: DOF of the Plastic Hinge Beam Element Showing Nodal Translations and Rotations and Nodal Forces and Moments

The major and minor principal axes, Y' and Z', of the cross-section are oriented using a user-defined orientation angle, α_{or}, that is defined differently for beams and columns as described below.

For columns, α_{or} is the angle made by the Z' axis with a plane passing through the longitudinal axis of the element, X' and the global X axis (See Figure 4.1). The direction of α_{or} is defined using the right-hand-thumb rule and is positive if it is in the positive direction of X' axis. It is updated after every iteration during each time step of the analysis based on the twist (torsional deformation) in the element.

For beams, α_{or} is the angle made by the Z' axis with a plane passing through the longitudinal axis of the element, X' and the global Z axis (See Figure 4.1). When the global Z axis is the vertical axis, this plane is a global vertical plane passing through the longitudinal axis of the element. The direction of α_{or} is defined using the right-hand-thumb rule and is positive if it is in the positive direction of X' axis. It is updated after every iteration during each time step of the analysis based on the twist (torsional deformation) in the element.

3.2.1 Degrees of Freedom, and Nodal Forces and Moments

The degrees of freedom (Figure 3.4) of the plastic hinge element are:

1. U_1, U_2 = X' translations at nodes 1 and 2 respectively.

2. $V_{1Y'}, V_{2Y'}$ = Y' translations at nodes 1 and 2 respectively.

3. $V_{1Z'}, V_{2Z'}$ = Z' translations at nodes 1 and 2 respectively.

4. α_1, α_2 = Rotations of the element about X' at nodes 1 and 2 respectively.

5. $\theta_{1Y'}, \theta_{2Y'}$ = Rotations of the element about Y' at nodes 1 and 2 respectively.

6. $\theta_{1Z'}, \theta_{2Z'}$ = Rotations of the element about Z' at nodes 1 and 2 respectively.

Corresponding to these DOF are nodal forces and moments (Figure 3.4):

1. P_1, P_2 = Forces in X' direction at nodes 1 and 2 respectively.

2. $Q_{1Y'}, Q_{2Y'}$ = Forces in Y' direction at nodes 1 and 2 respectively.

3. $Q_{1Z'}, Q_{2Z'}$ = Forces in Z' direction at nodes 1 and 2 respectively.

4. T_1, T_2 = Moments about X' axis at nodes 1 and 2 respectively.

5. $M_{1Y'}, M_{2Y'}$ = Moments about Y' axis at nodes 1 and 2 respectively.

6. $M_{1Z'}, M_{2Z'}$ = Moments about Z' axis at nodes 1 and 2 respectively.

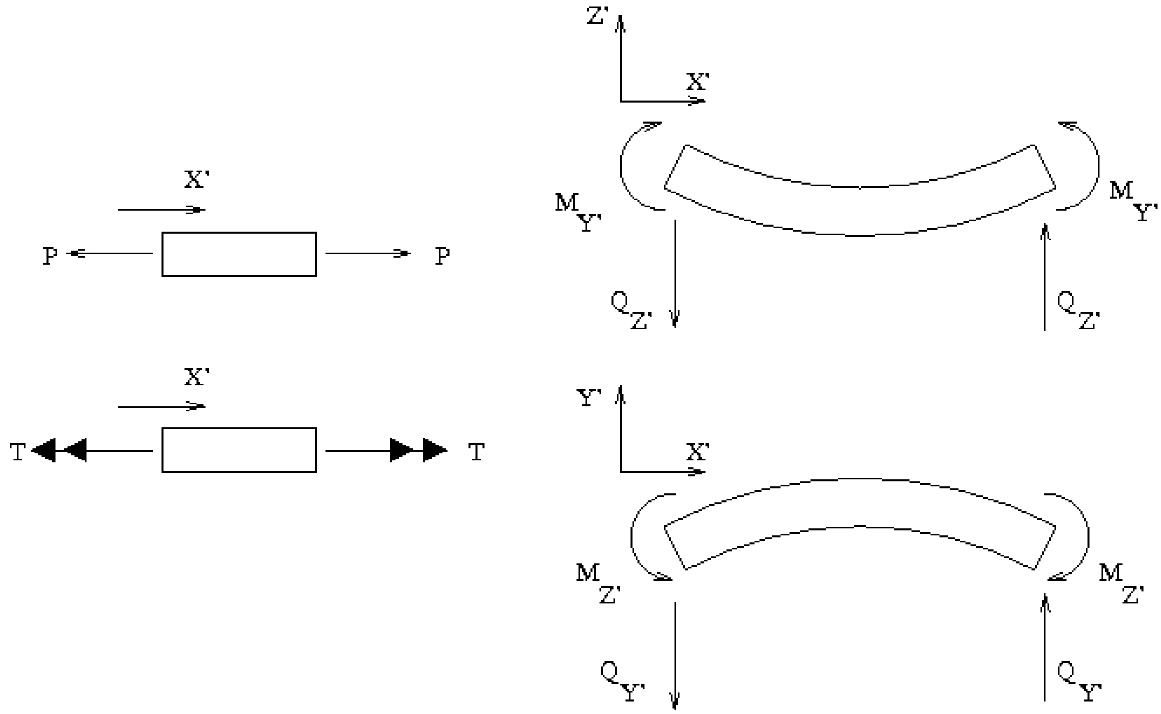


Figure 3.5: Sign Convention for Internal Forces and Couples in a Plastic Hinge Beam Element

3.2.2 Internal Forces and Moments

The set of internal forces and moments in the plastic hinge beam element are as follows:

1. P = Axial force in the element.
2. $Q_{Y'}$, $Q_{Z'}$ = Shear forces in the element in the Y' and Z' directions respectively.
3. T = Twisting moment in the element.
4. $M_{Y'}$, $M_{Z'}$ = Moments in the element about the Y' and Z' axes respectively.

The sign convention (positive directions) for these forces and moments is shown in Figure 3.5.

3.2.3 Material Nonlinearity

Two types of material nonlinearity are considered.

1. Axial yielding when axial force, P, in the element reaches the yield axial force, P_y, which is given by:

$$P_y = \sigma_y A \tag{3.6}$$

where σ_y is the yield stress of the material and A is the area of the cross-section.

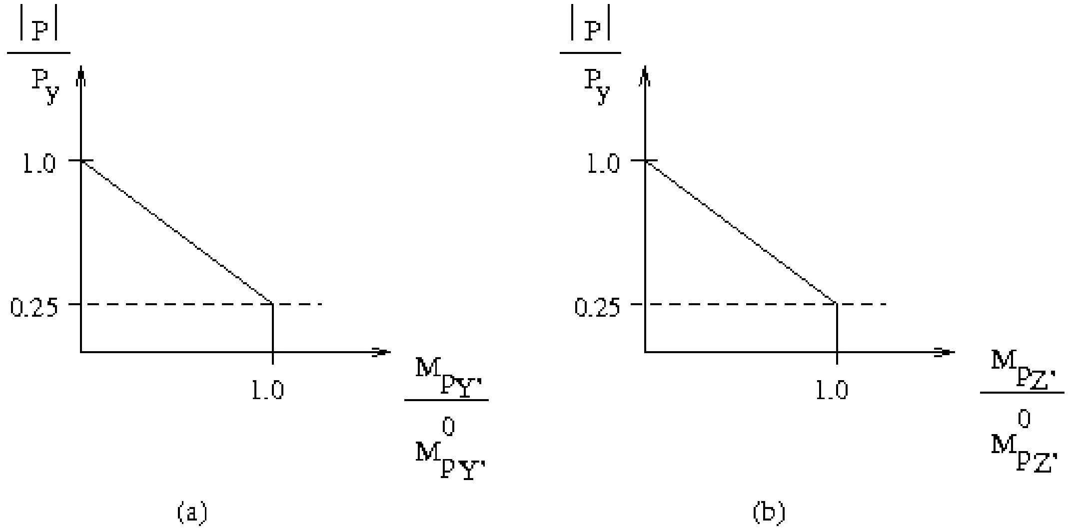


Figure 3.6: $P - M_{pY'} - M_{pZ'}$ Interaction Diagrams used for the Plastic Hinge Element

2. Flexural yielding when the bending moment, $M_{Y'}$ or $M_{Z'}$ at the ends of the element reaches the plastic moment capacity, $M_{pY'}$ or $M_{pZ'}$. Two PMM interaction relations are incorporated. The first relation, shown in Figure 3.6, accounts for the dependency of the major and minor plastic moment capacities on the axial force, but not on the moments in either direction. Thus, the computed plastic moment capacity in one direction is independent of the moment in the other direction, and vice-versa. In this case, given the axial force in the member, the plastic moment capacity in either direction is calculated using the relations shown in Figure 3.6. The plastic moment capacities when $P = 0$, denoted by $M^0_{pY'}$ and $M^0_{pZ'}$, are given by

$$M^0_{pY'} = \sigma_y Z_{Y'} \tag{3.7a}$$

$$M^0_{pZ'} = \sigma_y Z_{Z'} \tag{3.7b}$$

The second relation is a discretized interaction surface accounting for dependency of the plastic moment capacities in either direction not only on the axial force, but also on the moments in the two directions. Examples of normalized surfaces for wide-flange and tube sections are shown in Figure 3.7. A radial vector originating at the origin and passing through the current state of the plastic hinge element represented by the point $(P, M_{Y'}, M_{Z'})$ is extended to intersect the interaction surface. The axial force and minor & major axis bending moments at this point of intersection are taken to be the capacities. If the current state lies outside the PMM surface, the strain-hardening moments are removed from the current state and the radial vector is recomputed and extended to meet the yield surface to determine the axial and moment capacities. There is one limitation, however. The normalized surface corresponding to a single user-specified section from the model is used for all cross-sections in the model. where $Z_{Y'}$ and $Z_{Z'}$ are the plastic moduli of the cross-section of the element about its major and minor axes respectively. Once $|M_{Y'}|$ or $|M_{Z'}|$ reaches $|M_{pY'}|$ or $|M_{pZ'}|$ at node 1 or node 2, further loading causes a kink, termed a plastic hinge to form in the beam at that node. Between these plastic hinge locations, the beam behaves elastically. To approximate the effect of strain hardening, elastic rotational springs are mounted across the plastic hinges to exert moments proportional to the kink angles about Y' and Z' axes.

3.3 Elastofiber Beam Element

The following assumptions are made in the formulation of this element:

1. The cross-section is assumed uniform along the length of the element.
2. The cross-section is assumed to be doubly symmetric (with shear center at the centroid).
3. Plane sections remain plane; however, they do not have to remain normal to the beam axis. Thus, shear deformations are included.
4. Strains in the element are small.
5. Lateral deflections relative to the chord are small. This means that the effect of bowing (Oran 1973a; Oran 1973b; Kassimali 1983) on axial stiffness is neglected. This effect is important only for very slender members or during post-buckling of compression members such as braces in braced frames.
6. Warping restraint under twisting is neglected.
7. The element is loaded only at its ends.
8. Composite action with the slab is not included.
9. Local buckling is not included.

The elastofiber beam element has 3 segments and 4 nodes. Two nodes numbered 1 and 2 are located at the ends and connect to the attachment points a through f of the panel zone element. The other two nodes are interior and are numbered 3 and 4 (See Figure 3.8. This element can model beams and columns in framed structures. As in the case of the plastic hinge element, columns connect to attachment points e and/or f while beams connect to attachment points a through d. Original length of a segment is denoted by L_{s0}. Segment 1 goes from node 1 to node 3, segment 2 from node 3 to node 4, and segment 3 from node 4 to node 2.

Each segment has its own local coordinate system, $X'Y'Z'$, which is a right-handed, orthogonal system and is defined as follows:

1. X' axis: It runs along the longitudinal axis of the segment at the centroid of the cross-section. It is defined as a vector from node 1 to node 3 for segment 1, from node 3 to node 4 for segment 2, and from node 4 to node 2 for segment 3. All the nodes are located at the centroid of the cross-section.

2. Y' axis: Y' axis is orthogonal to X' and is the major principal axis of the cross-section of the element.

3. Z' axis: Z' axis is the minor principal axis of the cross-section of the element.

Initially, beam element is straight and each of the three $X'Y'Z'$ systems has the same orientation. Y' and Z' axes are oriented in the same way as for the plastic hinge element using the orientation angle, α_{or}, defined differently for beams and columns as described in Section 3.2. The matrix of direction cosines, $[C']$, is computed as described in those sections for segments in beams and columns.

3.3.1 Degrees of Freedom of a Segment, and Nodal Forces and Moments

The degrees of freedom (Figure 3.9) of a segment of the elastofiber element are:

1. U_i, U_j = X' translations at nodes i and j of the segment respectively.

2. $V_{iY'}$, $V_{jY'}$ = Y' translations at nodes i and j respectively.

3. $V_{iZ'}$, $V_{jZ'}$ = Z' translations at nodes i and j respectively.

4. α_i, α_j = Rotations of the segment about X' at nodes i and j respectively.

5. $\theta_{iY'}$, $\theta_{jY'}$ = Rotations of the segment about Y' at nodes i and j respectively.

6. $\theta_{iZ'}$, $\theta_{jZ'}$ = Rotations of the segment about Z' at nodes i and j respectively.

Corresponding to these DOF are nodal forces and moments (Figure 3.9):

1. P_i, P_j = Forces in X' direction at nodes i and j respectively.

2. $Q_{iY'}$, $Q_{jY'}$ = Forces in Y' direction at nodes i and j respectively.

3. $Q_{iZ'}$, $Q_{jZ'}$ = Forces in Z' direction at nodes i and j respectively.

4. T_i, T_j = Moments about X' axis at nodes i and j respectively.

5. $M_{iY'}$, $M_{jY'}$ = Moments about Y' axis at nodes i and j respectively.

6. $M_{iZ'}$, $M_{jZ'}$ = Moments about Z' axis at nodes i and j respectively.

3.3.2 Internal Forces and Moments in a Segment

The set of internal forces and moments in a segment of the elastofiber element are as follows:

1. P = Axial force in the segment.

2. $Q_{Y'}$, $Q_{Z'}$ = Shear forces in the segment in the Y' and Z' directions respectively.

3. T = Twisting moment in the segment.

4. $M_{Y'}$, $M_{Z'}$ = Moments in the segment about the Y' and Z' axes respectively.

The sign convention (positive directions) for these forces and moments is shown in Figure 3.5.

The middle segment of the elastofiber beam element is an elastic version of the plastic hinge beam element, i.e., no axial yielding and no plastic hinging. End segments are fiber segments. As shown in Figure 3.8, fiber segment cross-sections are divided into N fibers; each fiber runs the length of the segment. Associated with each fiber is a nonlinear hysteretic stress-strain law for axial stress, σ_n, and axial strain, ϵ_n, where n denotes the n^{th} fiber. This accounts for nonlinear coupling between bending about the Y' (major) and Z' (minor) axes, and axial deformation.

The fiber segment is based on finite element methodology whereby the beam translations and rotations are interpolated linearly and independently from the nodal values. This requires a one-point integration on the shear terms to prevent locking.

Because of the presence of the interior nodes, the updating process for each elastofiber beam element requires a nonlinear structural analysis. Iterations for each element are performed within each global iteration. The multi-segment element analyses are performed with degrees of freedom transformed to the global coordinate system, XYZ.

3.4 Material Model for Fiber Axial Stress-Strain

The axial stress-strain behavior of the fiber is governed by the hysteretic uniaxial cubic-ellipsoidal law proposed by Hall and Challa (1992, 1995). The stress-strain curve for monotonic loading, shown in Figure 3.10, is symmetric about both the strain and stress axes. It consists of an elastic portion, a yield plateau and a strain-hardening region which is described by a cubic ellipse. The monotonic stress-strain curve is characterized by seven parameters: the yield stress, σ_y, the ultimate stress, σ_u, the yield strain, ϵ_y, the strain at initiation of strain hardening, ϵ_{sh}, the strain at ultimate stress, ϵ_u, the rupture strain, ϵ_r, and the tangent modulus at the initiation of strain hardening, E_{sh}.

The cubic ellipse describing the strain hardening region is assumed to be centered at $(\epsilon_0 = \epsilon_u, \sigma_0)$ with radii, a and b:

$$\frac{(\epsilon_n - \epsilon_0)^3}{a^3} + \frac{(\sigma_n - \sigma_0)^3}{b^3} = 1 \tag{3.8}$$

Parameters, σ_0, a, and b, are determined from the user-defined properties, ϵ_{sh}, ϵ_u, E_{sh}, σ_y, and σ_u.

The hysteresis laws required to describe the stress-strain behavior under cyclic loading or reversal of loading are described in Challa (1992) and Hall and Challa (1995). Sample hysteresis loops are shown in Figure 3.11.

3.5 3-D Modified Elastofiber (MEF) Element to Model Slender Columns and Braces

While the elastofiber element is able to accurately capture nonlinear behavior at the ends of beams and columns, capturing nonlinear behavior (buckling, cracking, and rupture) at the middle of braces and slender columns requires the insertion of an additional central fiber segment. This modification to the elastofiber element leads to a 5-segment element with three fiber segments, two at the ends and one in the middle (Figure 3.12), separated by two interior segments modeled as elastic plastic hinge elements, i.e., with no axial yielding or plastic hinging. The three fiber segments are divided into 20 fibers in the cross-section, with each fiber running the full length of the segment. The axial stress-strain behavior of each fiber is governed by the nonlinear axial stress (σ_n) – strain (ϵ_n) law used to describe the behavior of fibers in the elastofiber element (Figures 3.10 and 3.11). n denotes the n^{th} fiber. Assumptions in the modified elastofiber (MEF) element formulation include prismatic doubly-symmetric sections, plane sections remain plane, small strains, no warping restraint, and no along-span loads. Lateral deflections relative to the chord in the two elastic segments are assumed small. Each of the six nodes of the MEF element have 6 degrees of freedom, three translational and three rotational. The interior nodes are assumed massless, and this allows for static condensation to be performed on the associated DOF, labeled 1–24 in Figure 3.12.

The segment degrees of freedom, and internal forces & moments are identical to the elastofiber element. Greater details of the MEF element can be found in Krishnan (2009).

3.6 Elastic 4-Noded Diaphragm Element

Floor slabs in buildings act as diaphragms that tie the moment frames together such that they act as one unit. In FRAME3D, the floor slabs are modeled using 4-noded elastic diaphragm elements. The diaphragm element is a 4-noded linear-elastic plane element that can take a quadrilateral form in 3D spatial coordinates, XYZ. It is used to provide translational restraint caused by floor slabs to columns in buildings. The following assumptions are made in the formulation of this element:

1. Material is isotropic and linear-elastic, with Young's modulus, E, shear modulus, G, and Poisson's ratio, ν.

2. Warping is not considered.

3. A state of plane-stress is assumed in the element.

4. The element continues to provide the same restraint (in magnitude and direction) as in the original configuration even though during the course of the analysis the nodes of the element may have translated causing the geometry of the element to change.

5. The element is not externally loaded and all the stresses in the element are generated by stiffness forces due to the translation of the nodes during the course of the analysis.

The nodes of the element are labeled i, j, m, and n. Each element has its own local coordinate system, $X'Y'$, which lies in the plane of the element and is defined as follows:

1. X' is in the direction of a vector from node j to node k.

2. Y' is orthogonal to X' and lies in the plane of the 4 nodes.

3.6.1 Degrees of Freedom, and Nodal Forces

The degrees of freedom (Figure 3.13) of a diaphragm element are:

1. $V_{iX'}$, $V_{jX'}$, $V_{mX'}$, $V_{nX'}$ = X' translations at nodes i, j, m, and n of the element respectively.
2. $V_{iY'}$, $V_{jY'}$, $V_{mY'}$, $V_{nY'}$ = Y' translations at nodes i, j, m, and n of the element respectively.

Corresponding to these degrees of freedom are nodal forces:

1. $Q_{iX'}$, $Q_{jX'}$, $Q_{mX'}$, $Q_{nX'}$ = X' forces at nodes i, j, m, and n of the element respectively.
2. $Q_{iY'}$, $Q_{jY'}$, $Q_{mY'}$, $Q_{nY'}$ = Y' forces at nodes i, j, m, and n of the element respectively.

3.6.2 Internal Stresses in a Diaphragm Element

The internal stresses in the diaphragm element are:

1. $\sigma_{X'X'}$ = Normal stress in X' direction.
2. $\sigma_{Y'Y'}$ = Normal stress in Y' direction.
3. $\sigma_{X'Y'}$ = Shear stress in the element.

These can be transformed to principal tensile and compressive stresses, σ_1 and σ_2 respectively, using Mohr's circle.

3.7 Elastic Translational or Rotational Spring Element

The program includes elastic translational and rotational spring elements. Springs can be used to model foundations. They can also be used to retrieve reaction forces and moments at the supports. Springs must be oriented in one of the three directions of the global coordinate system, XYZ.

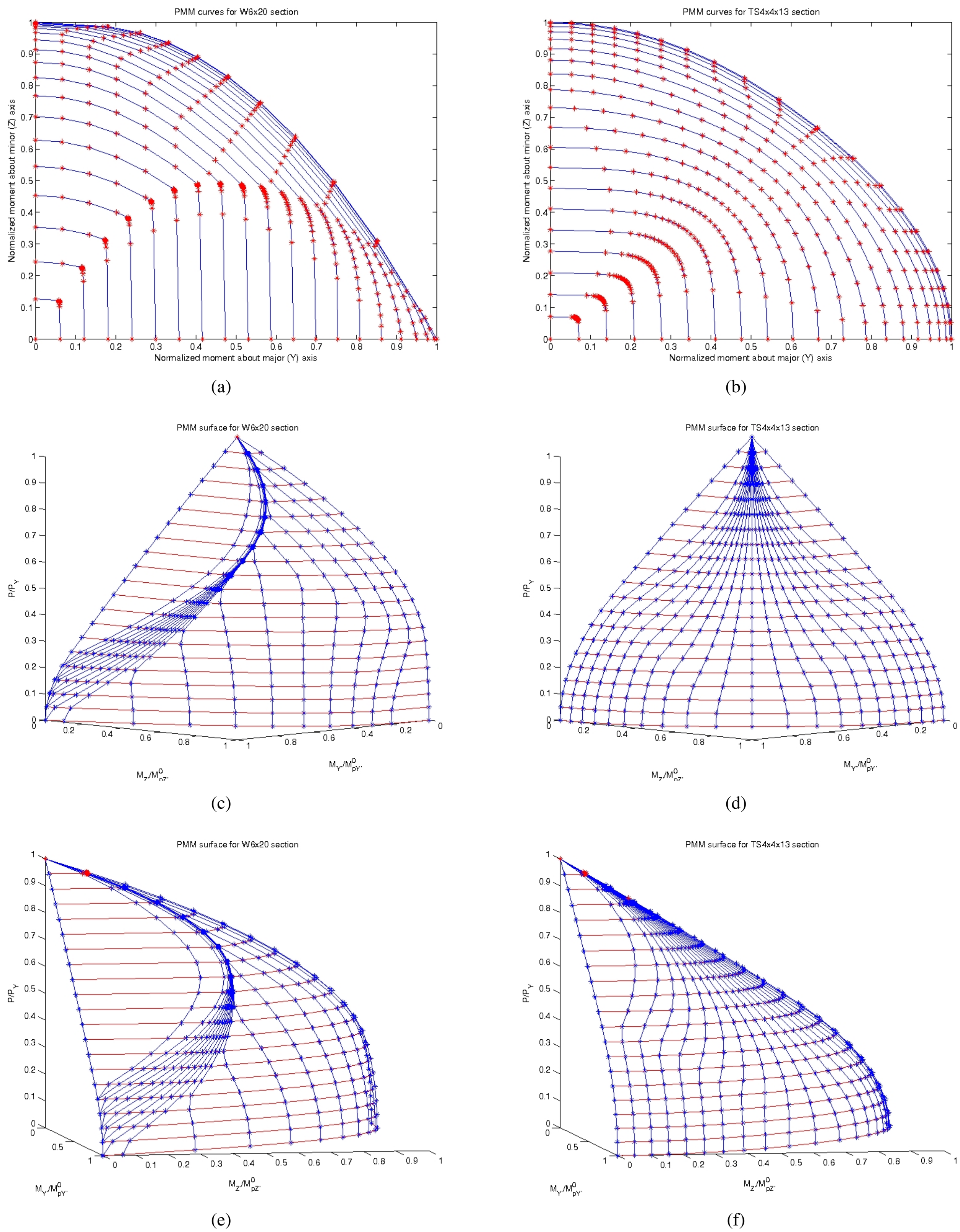


Figure 3.7: PMM interaction surfaces for W6x20 ((a), (c), & (e)) and TS4x4x13 ((b), (d), & (f)) sections

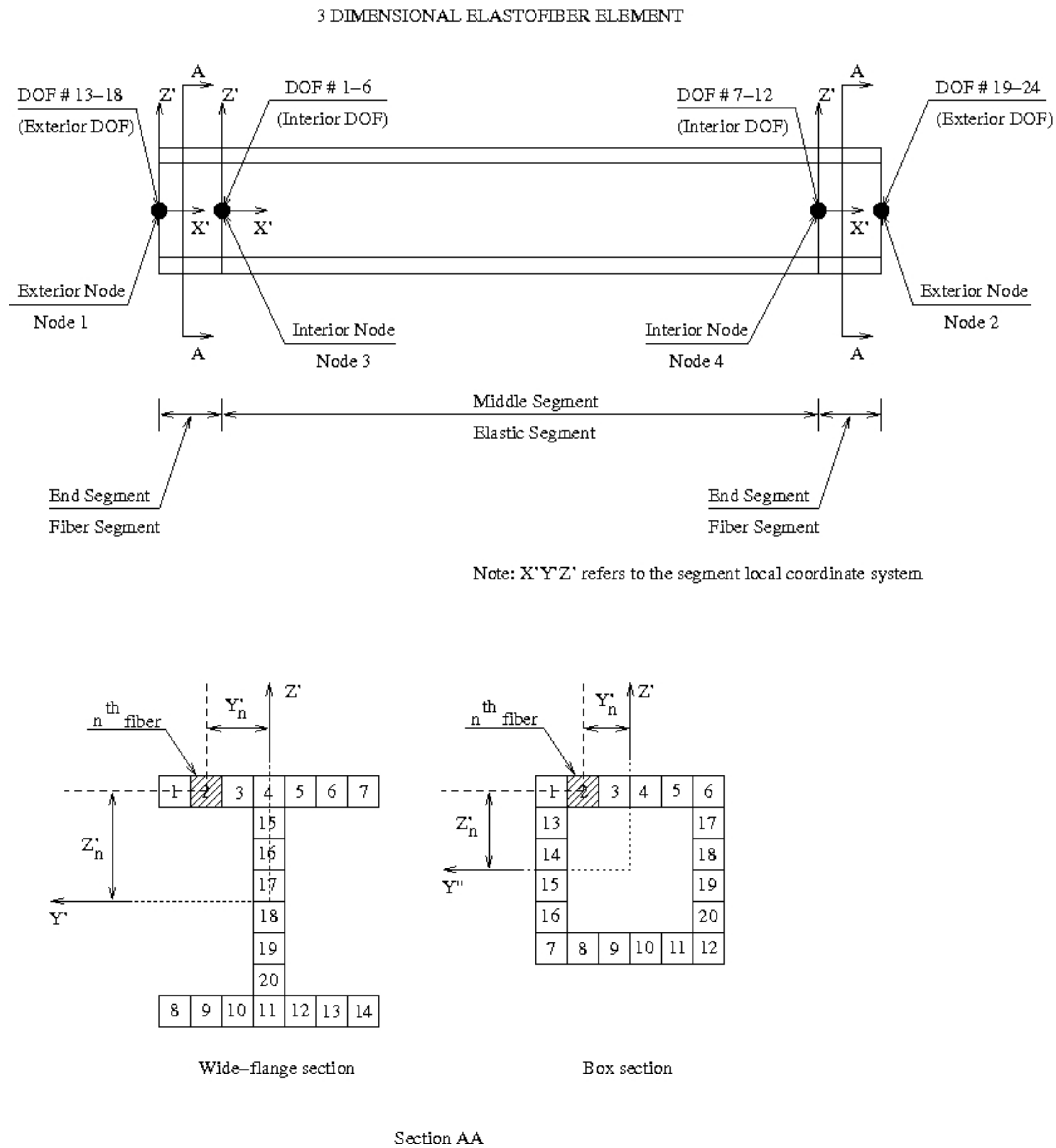


Figure 3.8: Layout of the 3-Segment Elastofiber Element

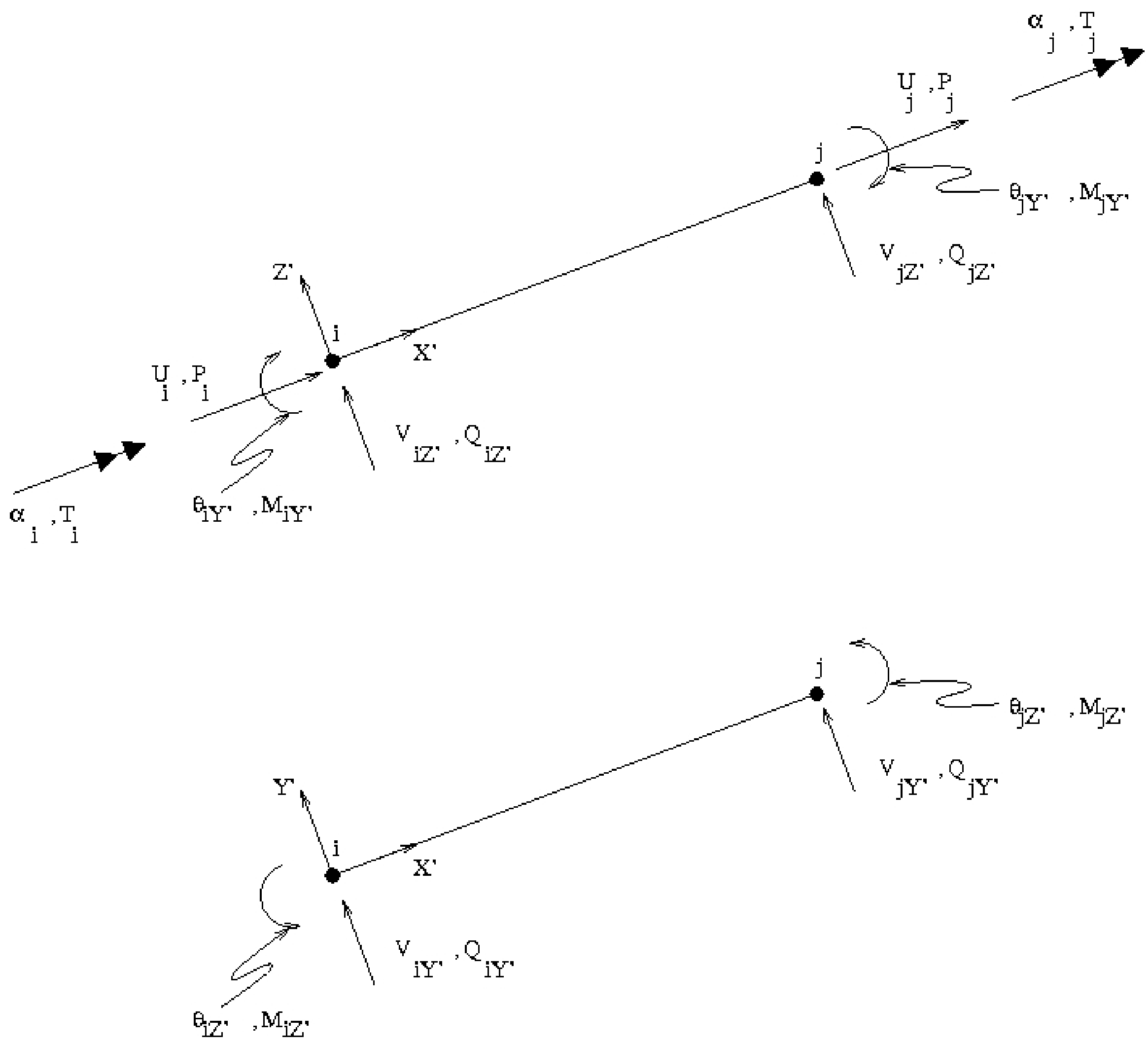


Figure 3.9: DOF of a Segment of the Elastofiber Beam Element Showing Nodal Translations and Rotations and Nodal Forces and Moments

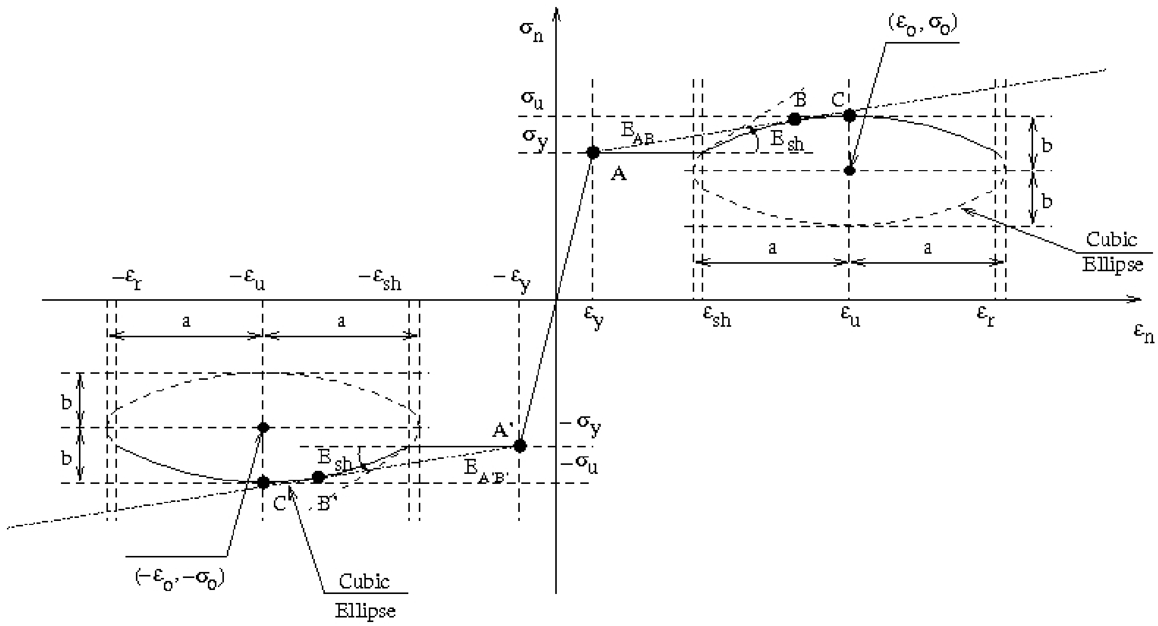


Figure 3.10: Backbone Stress-Strain Curve for a Typical Fiber in a Fiber Segment of an Elastofiber Element

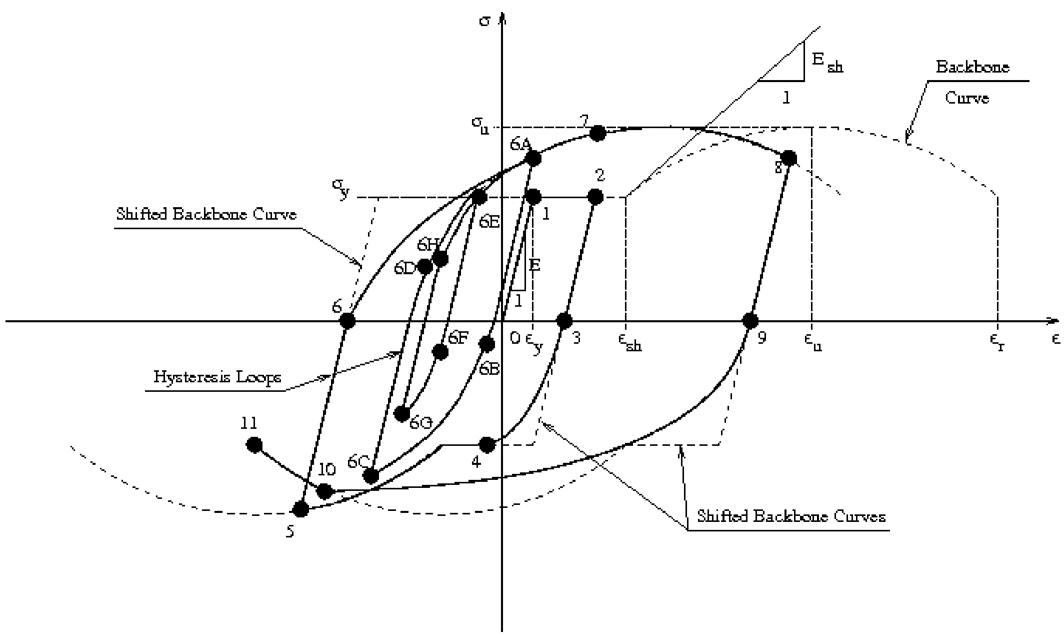


Figure 3.11: Axial stress-strain hysteresis model for each fiber.

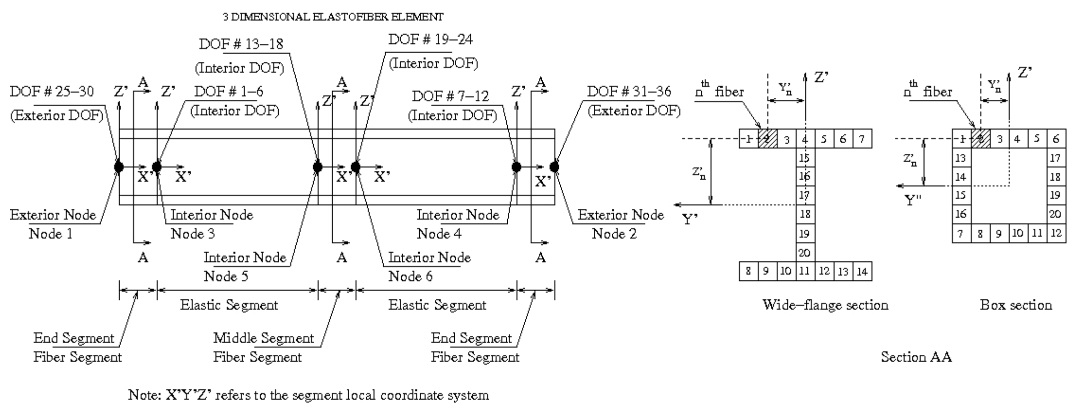


Figure 3.12: Layout of the five-segment modified elastofiber element (fiber arrangement is shown for an I-section and a box section).

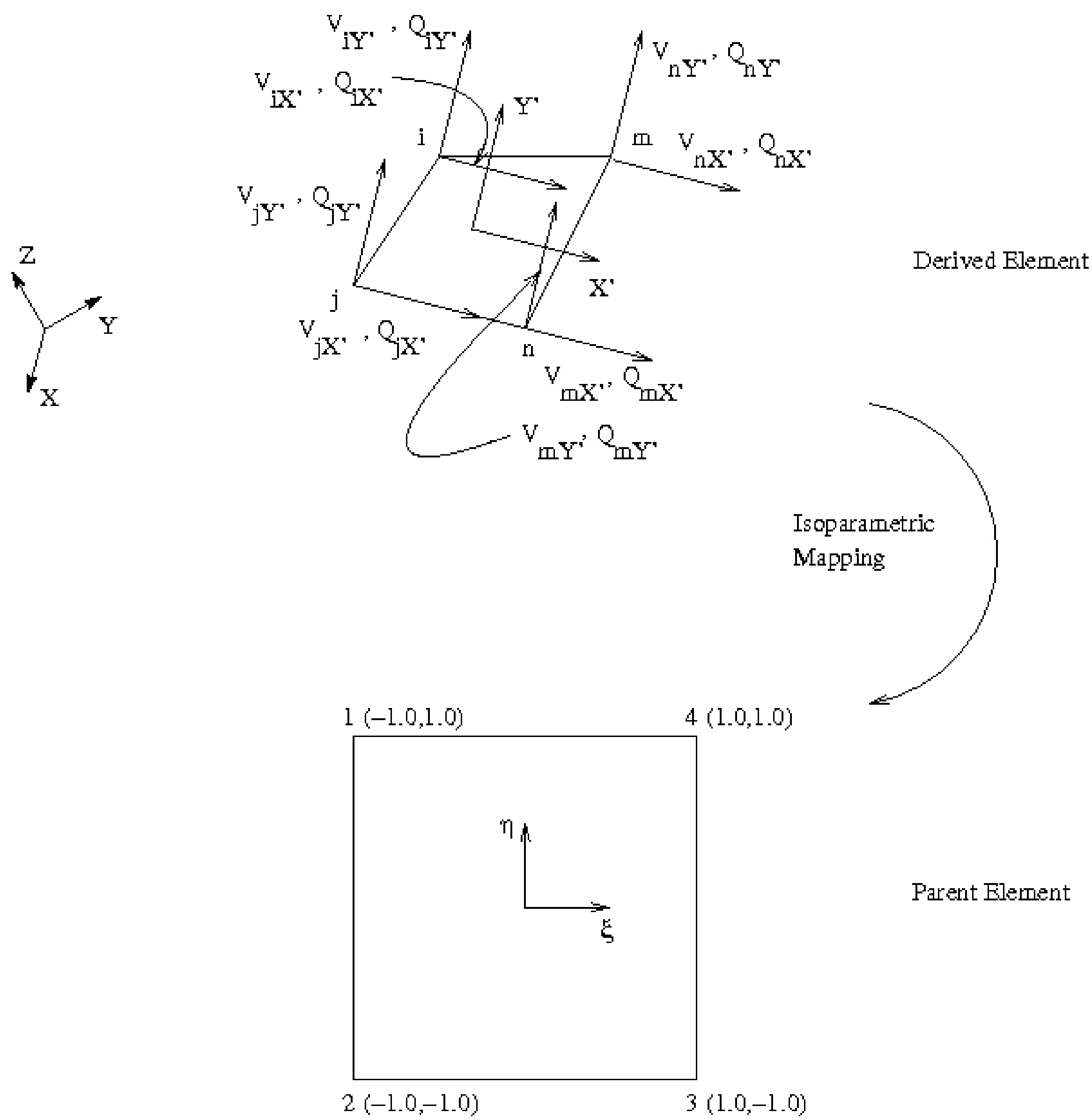


Figure 3.13: DOF of the Diaphragm Element Showing Nodal Translations and Rotations and Nodal Forces and the Isoparametric Mapping from a Square Parent Element

Chapter 4 Input Data

Five input files are required. These are listed in Table 4.1.

Table 4.1: Input Files for FRAME3D Analysis

File Name	Description
$for005$	Structural model input file.
$for009$	Section property database.
$for090$	Global X direction acceleration record.
$for091$	Global Y direction acceleration record.
$for092$	Global Z direction acceleration record.

4.1 Input File for005

Structure input should be prepared in a file named “for005”. All input data can be prepared in free format. There are 17 data blocks each separated by at least one comment/blank line (the actual number of comment/blank lines are given in Table 4.2). Note that the comment/blank lines are always required even if there’s no data required in a certain data block.

The following subsections detail the data format for the input file, $for005$. Each subsection is divided into three parts:

1. Subsubsection, “Format”, gives the sequence of entries to be provided in each data line of the file. Each entry corresponds to a single variable.

2. Subsubsection, “Description”, gives a tabular description in the form:

   Variable Field Note Definition

   Variable is the name of the variable/property.

   Field is the locator of the **Variable** in the current data line. Thus, if the current **Variable** is the third number on the current data line, then, **Field** would be 3.

   Note is the reference to the note relevant to the current **Variable** in Subsubsection “Notes”.

   Definition gives a short description of the **Variable**.

3. Subsubsection, “Notes”, gives a list of notes describing the **Variable** in greater detail.

Table 4.2: File for005: List of Data Blocks

S.No.	Data Block	When Needed	No.of Comment/Blank Lines Preceding the Data Block
1.	Heading	Always	1
2.	Main Control Data	Always	1
3.	Integration and Iteration Data	Always	1
4.	Nodal Data	Always	1
5.	Spring Element Data	Only if $NSPR > 0$	1
6.	Beam Element Data	Only if $NPHEL > 0$ or $NFIBEL > 0$	1
7.	Categories for Elastofiber Element Fiber Area Reduction	Always	1
8.	Seed for Random Number Generation for Elastofiber Element Fiber Fracture Strain Computation	Always	1
9.	Categories for Elastofiber Element Fiber Fracture Strain Prob. Distribution	Always	1
10.	Panel Zone Element Data	Only if $RYRES > 0$ or $RZRES > 0$ for at least one node	3
11.	Diaphragm Element Control Data	Always	1
12.	Diaphragm Element Material Data	Only if $NUMPLSTRMAT > 0$	1
13.	Diaphragm Element Connectivity Data	Always	4
14.	Diaphragm Element Output Control Data	Only if $NPLSTRSS > 0$	2
15.	Time History Output Control Data	Only if $NRTH > 0$	3
16.	Element Group Resultant Data	Only if $NGRP > 0$	3
17.	Ground Acceleration Scaling Factors	Always	1
18.	Deformed Shape Output Control Data	Always	1
19.	Yield Stress Units	Always	1
20.	Peak Interstory Drift Output Control Data	Always	1

4.1.1 Heading

Prepare one line of data up to 72 characters describing the model for output labeling.

4.1.2 Main Control Data

Format

Prepare one line of data to define the control parameters of the structural model in the following form:

NNP NPHEL NFIBEL NFIBEL5[1] NPZ0 NPZ1 NSPR MTP NDIM MBD NSSTPS NDSTPS NRTH NGITMAX NLITMAX NOUT ISEC NCURVES[1] NPCURVE[1] NGRP[1] NELMGRPMAX[1] NEIG[1]

Description

Variable	Field	Note	Definition
NNP	1	(1)	Number of node points.
$NPHEL$	2	(2)	Number of plastic hinge beam elements.
$NFIBEL$	3	(3)	Number of elastofiber beam elements.
$NFIBEL5$	4	(4)	Number of 5-segment modified elastofiber (MEF) elements.
$NPZ0$	5	(5)	Number of panel zones with bilinear shear stress-strain relation.
$NPZ1$	6	(6)	Number of panel zones with linear-quadratic shear stress-strain relation.
$NSPR$	7	(7)	Number of translational and rotational springs.
MTP	8	(8)	Maximum number of turning points.
$NDIM$	9	(9)	Storage parameter for turning point locations.
MBD	10	(10)	Global half bandwidth.
$NSSTPS$	11	(11)	Number of static analysis steps.
$NDSTPS$	12	(12)	Number of dynamic analysis steps.
$NRTH$	13	(13)	Number of response time histories to be output.
$NGITMAX$	14	(14)	Maximum number of global iterations per time step.
$NLITMAX$	15	(15)	Maximum number of local member (elastofiber element) iterations per time step.
$NOUT$	16	(16)	Output data is written for each $NOUT^{th}$ time step.
$ISEC$	17	(17)	Section to be used for PMM interaction surface computation.
$NCURVES$	18	(17)	Number of curves in the PMM interaction surface.
$NPCURVE$	19	(17)	Number of points in each curve comprising the PMM interaction surface.
$NGRP$	20	(18)	Number of element groups for which cumulative resultant forces are to be output.
$NELMGRPMAX$	21	(18)	Maximum number of elements in any of the $NGRP$ groups.
$NEIG$	22	(19)	Number of eigen values and vectors to be computed.

[1]Introduced in Version 2.0

Notes

1. This variable defines the number of nodal points required by the user to describe the structural geometry. There can then be a maximum of $NNP * 8$ degrees of freedom in the model.

2. This variable defines the number of beam elements that are to be modeled as plastic hinge elements with bilinear moment-rotation relation. Identify these elements in the beam element data block by setting the variable, $IELTYPE$, to 0.

3. This variable defines the number of beam elements that are to be modeled as elastofiber elements with cubic-ellipsoidal axial stress-strain relation for each fiber. Identify these elements in the beam element data block by setting the variable, $IELTYPE$, to 1.

4. This variable defines the number of beam elements that are to be modeled as 5-segment modified elastofiber elements with cubic-ellipsoidal axial stress-strain relation for each fiber. Identify these elements in the beam element data block by setting the variable, $IELTYPE$, to 2.

5. This variable defines the number of panel-zones that are to be modeled using a bilinear material shear stress-strain relation. Identify these panel zones in the panel zone element data block by setting the variable, $IPZTYPE$, to 0. Note that each panel zone element can have 2 mutually perpendicular panel zones. Data for each of these panel zones needs to be provided separately.

6. This variable defines the number of panel-zones that are to be modeled using a linear-quadratic material shear stress-strain relation. Identify these panel zones in the panel zone element data block by setting the variable, $IPZTYPE$, to 1. Note that each panel zone element can have 2 mutually perpendicular panel zones. Data for each of these panel zones needs to be provided separately.

7. This variable defines the number of elastic springs to be specified in the model. Springs can be used to model foundations. They can also be used to retrieve reaction forces and moments at the supports. In this case, the spring constant/stiffness (the variable, $SPRSTF$, in the Spring Element data block) should be specified to be much greater (at least four orders of magnitude greater) than the stiffness of the other elements in the model.

8. This variable is used to set the storage for coordinates of the turning points in the stress-strain history of each fiber of the elastofiber element. A reasonable value would be between 20 and 40.

9. This variable defines the dimension of the array storing turning point data for all fibers of all elastofiber elements in the model. This variable need not exceed $NFIBEL * 2 * 20 * MTP + NFIBEL5 * 3 * 20 * MTP$. It can be far less than this because most fibers in the structural model would not traverse through MTP turning points in their stress-strain history.

10. This variable is used to set the storage of the stiffness and damping matrices. These matrices are stored in banded form and the user needs to specify the maximum global half-bandwidth. The program itself

computes a half-bandwidth based on the structural geometry and degrees of freedom. If the user-specified MBD is less than that computed internally by the program, then program is terminated with a warning printed in the output file, $for006$.

11. This variable defines the number of steps in which the user-specified static loads in the Nodal data block are to be applied. These loads are divided by this number and this fraction is applied incrementally at each of the static load steps.

12. This variable defines the number of steps for which dynamic loads are to be applied to the structure. Dynamic loads can be specified by providing time-histories of three mutually orthogonal components of acceleration in input files $for090$, $for091$, and $for092$, that accelerate the nodal masses specified in the Nodal data block inducing dynamic inertial forces in the structure. $NDSTPS$ can be less than the number of entries in the acceleration records.

13. This variable defines the number of response time histories for which output data is required. The time-history data will be written out in two separate files $for006$ and RTH. Response quantities include nodal displacements, velocities, accelerations, rotations, spring forces, beam element forces, moments, plastic rotations, axial strain, joint moments, shear strains, and elastofiber/MEF element fiber stresses and strains. $NRTH$ is limited to 1000 due to output formatting constraints.

14. This variable defines the maximum number of global iterations to be implemented at each time-step for convergence to the correct solution. If convergence is not achieved within $NGITMAX$ iterations, a warning is printed out in the output file, $for006$, and analysis proceeds to the next time step. A value of 50 is recommended for this variable.

15. This variable defines the maximum number of local member iterations to be implemented for each elastofiber element at each global iteration of each time-step. If convergence is not achieved within $NLITMAX$ iterations, a warning is printed out in the output file, $for006$, and analysis proceeds to the next element in the current global iteration of the current time-step. A value of 1200 is recommended for this variable.

16. This variable controls the frequency of time-history output printing. If data is required to be written out at each time-step, then $NOUT$ should be assigned the value, 1. If data is required to be written out only every fourth time-step, then $NOUT$ should be assigned the value, 4.

17. $ISEC$ defines the section for which the normalized PMM interaction surface is computed. This discretized functional form is used in determining the instantaneous axial and bi-directional flexural capacities of all plastic hinge elements in the model under combined loading, in conjunction with the theoretical axial and flexural capacities under uniaxial loading (axial or minor axis flexure or major axis flexure, respectively) conditions. If there are tube sections in the model, it is preferable to use the normalized interaction surface corresponding to one of the tube sections. This would result in smaller errors. It is sufficient to constrain the surface using 21 curves and 21 points on each curve. Thus, $NCURVE$ and $NPCURVE$ can both be assigned a value of 21.

18. In structural models, it may be of interest to study the cumulative forces of a group of members. For example, to determine the story shear forces in any given story of a moment frame building, the shear forces in all the columns need to be added. In FRAME3D one can do this by grouping all the columns in a story into a group. The program will compute the sum of the 12 components of forces in all the members of the group. The forces in each column are first transformed to the global coordinate system, and the summation is performed thereafter. The force resultants are thus written out in the global coordinate system. The number of such groups and the maximum number of elements in any of the groups have to be specified, so that sufficient memory can be internally allocated.

19. A subspace iteration procedure (Bathe 1996) is used to determine the lowest $NEIG$ natural frequencies and corresponding mode shapes of the model. The output is presented in two files MODES and EIGEN.

4.1.3 Integration and Iteration Data

Format

Prepare one line of data to define the control parameters for integration and iteration during the solution process in the following form:

DT TIMEOSTF BETA GAMMA A0 A1 AGRAV STFFACPHE STFFACFE STFFACPZ0 STFFACPZ1 TOLF TOLM TOLFIBELF TOLFIBELM THRESH NTHR ITHR

Description

Variable	Field	Notes	Definition
DT	1	(1)	Time increment for dynamic time-history analysis.
$TIMEOSTF$	2	(2)	Time till which the initial elastic global stiffness matrix is to be used for Newton-Raphson iteration.
$BETA$	3	(3)	Newmark integration parameter.
$GAMMA$	4	(3)	Newmark integration parameter.
$A0$	5	(4)	Rayleigh damping parameter.
$A1$	6	(4)	Rayleigh damping parameter.
$AGRAV$	7	(5)	Acceleration due to gravity.
$STFFACPHE$	8	(6)	Fraction of elastic stiffness of plastic hinge element to be added to its tangent stiffness for Newton-Raphson iteration.
$STFFACFE$	9	(7)	Fraction of elastic stiffness of an elastofiberor MEF element to be added to its tangent stiffness for Newton-Raphson iteration.
$STFFACPZ0$	10	(8)	Fraction of elastic stiffness of bilinear panel zone element to be added to its tangent stiffness for Newton-Raphson iteration.
$STFFACPZ1$	11	(9)	Fraction of elastic stiffness of linear-quadratic panel zone element to be added to its tangent stiffness for Newton-Raphson iteration.
$TOLF$	12	(10,14)	Force tolerance for global convergence of displacement DOF.
$TOLM$	13	(11,14)	Moment tolerance for global convergence of rotation DOF.
$TOLFIBELF$	14	(12,14)	Force tolerance for local convergence of displacement DOF of an elastofiber or MEF element.
$TOLFIBELM$	15	(13,14)	Moment tolerance for local convergence of rotation DOF of an elastofiber or MEF element.
$THRESH$	16	(15)	Threshold value for system picture.
$NTHR$	17	(15)	Threshold node for system picture.
$ITHR$	18	(15)	Threshold DOF for system picture.

Notes

1. This variable defines the time increment for dynamic loading. The acceleration records specified in files $for090$, $for091$, and $for092$, should consist of data points that are spaced at DT time units.

2. This variable defines the time until which the initial elastic global stiffness matrix is to be used for Newton-Raphson iteration. The Newton-Raphson method requires that the tangent stiffness matrix be generated and factored for equation solving in every iterative cycle. This is expensive. Thus, the user can pick an alternate strategy of using the initial stiffness matrix to iterate for a specified number of time steps and then switching to the tangent matrix formulation. This way, for the initial portion of the ground motion time-history, when the structure is still elastic or has just entered the nonlinear regime, huge computation time savings can be achieved, since the constant initial stiffness matrix needs to be factored only once. Of course, the solution process would require a larger number of iterations for convergence during these time steps but this is more than offset by the savings from matrix-factoring. If convergence with the initial stiffness matrix is too slow, try using the tangent stiffness matrix.

   In some cases, the solution using the tangent stiffness matrix may not converge, but could keep cycling back-and-forth between two values. Using the initial stiffness matrix may help to converge to the solution without getting lost, under these circumstances.

3. Newmark's method is unconditionally stable if $\gamma \geq 0.5$ and $\beta \geq \frac{(2\gamma+1)^2}{16}$. Artificial positive damping is introduced if $\gamma > 0.5$ and artificial negative damping if $\gamma < 0.5$. If $\gamma = 0.5$ and $\beta = 0$, Newmark's method reduces to the central difference method. A good choice of parameters which makes this method unconditionally stable in linear problems is $\gamma = 0.5$ and $\beta = 0.25$. In this case, Newmark's method reduces to the constant average acceleration method or the trapezoidal method. With these values for γ and β there are no amplitude errors in any sine-wave motion, regardless of its frequency. But periods of sine-wave motion are overestimated. The error increases as $\frac{\Delta t}{T}$ increases, where T is the correct period of the motion. Unconditional stability in linear problems however, is no guarantor for the algorithm to remain unconditionally stable when used to solve nonlinear problems.

4. The program assumes that the structural system exhibits Rayleigh damping, i.e. mass and stiffness proportional. Thus, the damping matrix is computed as

   $$[D] = a_0\,[M] + a_1\,[K] \tag{4.1}$$

   where a_0 and a_1 are user defined proportionality constants. The following discussion can be of help to the user in selecting reasonable values for a_0 and a_1.

   In the linear elastic case, where modal superposition is valid, if Rayleigh damping is assumed, the modal damping in the i^{th} mode can be written in terms of the corresponding modal frequency as

   $$\zeta_i = \frac{1}{2}\left(a_0\omega_i + \frac{a_1}{\omega_i}\right) \tag{4.2}$$

Based on this relation, if we plot ω_i vs ζ_i, the curve is found to be reasonably flat in the intermediate frequency range. Thus, assuming that the structure exhibits approximately equal damping values in all the modes, two frequencies, ω and $R\omega$, are picked covering the frequency range of interest and a desired amount of damping. Then a_0 and a_1 can be computed as

$$a_0 = \frac{2\zeta\omega R}{1+R} \tag{4.3a}$$

$$a_1 = \frac{2\zeta}{\omega\,(1+R)} \tag{4.3b}$$

5. This variable defines the acceleration due to gravity and is used to convert the nodal weights specified in the Nodal data block to nodal masses. If nodal masses are specified in the nodal data block instead of weights, then $AGRAV$ should be given a value of 1.0.

6. This variable defines the fraction of the elastic stiffness matrix to be added to the tangent stiffness matrix of all plastic hinge beam elements for iterating in order to keep the stiffness matrix from becoming ill-conditioned. A reasonable value for this would be between 0.01 and 0.02.

7. This variable defines the fraction of the elastic stiffness matrix to be added to the tangent stiffness matrix of all elastofiber and MEF beam elements for iterating in order to keep the stiffness matrix from becoming ill-conditioned. A reasonable value for softening systems would be between 0.01 and 0.02. For stiffening systems (e.g., braced frame systems where post-buckling straightening of braces during a tension excursion under cyclic loading loading may result in the system stiffening up), may need to use 0.1 or larger. **Warning: As elastofiber/MEF element fibers start rupturing, behavior becomes highly nonlinear. Member local iterations may not converge. If the local iterations for any member do not converge, a warning is issued in output file $for006$ and the analysis proceeds without termination. However, solution may be erroneous, especially after a few fibers have ruptured. For accurate solution, user will have to try various values of $STFFACFE$ until convergence is achieved in every member. Raising the convergence tolerance limit $TOLFIBELF$ might help.**

8. This variable defines the fraction of the elastic stiffness matrix to be added to the tangent stiffness matrix of all bilinear panel zone elements for iterating in order to keep the stiffness matrix from becoming ill-conditioned. A reasonable value for this would be between 0.01 and 0.02.

9. This variable defines the fraction of the elastic stiffness matrix to be added to the tangent stiffness matrix of all linear-quadratic panel zone elements for iterating in order to keep the stiffness matrix from becoming ill-conditioned. A reasonable value for this would be between 0.01 and 0.02.

10. This variable defines the displacement degree of freedom global convergence tolerance on force. If the unbalanced forces corresponding to all the displacement degrees of freedom are less than $TOLF$ and the unbalanced moments corresponding to all the rotational degrees of freedom are less than $TOLM$, the global iteration is deemed to have converged and analysis proceeds to the next time step.

11. This variable defines the rotational degree of freedom global convergence tolerance on moment. If the unbalanced forces corresponding to all the displacement degrees of freedom are less than $TOLF$ and the unbalanced moments corresponding to all the rotational degrees of freedom are less than $TOLM$, the global iteration is deemed to have converged and analysis proceeds to the next time step.

12. This variable defines the displacement degree of freedom member local iteration convergence tolerance on force (applicable to elastofiber and MEF elements). If the unbalanced forces corresponding to all the displacement degrees of freedom are less than $TOLFIBELF$ and the unbalanced moments corresponding to all the rotational degrees of freedom are less than $TOLFIBELM$, the member local iteration is deemed to have converged and analysis proceeds to the next MEF/elastofiber element in the current global iteration of the current time step.

13. This variable defines the rotational degree of freedom member local iteration convergence tolerance on moment (applicable to elastofiber and MEF elements). If the unbalanced forces corresponding to all the displacement degrees of freedom are less than $TOLFIBELF$ and the unbalanced moments corresponding to all the rotational degrees of freedom are less than $TOLFIBELM$, the member local iteration is deemed to have converged and analysis proceeds to the next MEF/elastofiber element in the current global iteration of the current time step.

14. For smooth convergence, the tolerance limits for convergence criteria on local member forces and moments ($TOLFIBELF$ and $TOLFIBELM$) must be an order of magnitude smaller than those on the forces and moments associated with the global degrees of freedom ($TOLF$ and $TOLM$). When using plastic hinge elements and MEF/ elastofiber elements simultaneously in a model, convergence problems can arise due to an improper choice of convergence tolerance limits.

15. The variable, $THRESH$, defines the threshold value of the degree of freedom (1-8), $ITHR$, of node, $NTHR$, beyond which the system picture is printed out in file, $for006$. The system picture consists of element forces, plastic rotations and stresses, and nodal displacements. If $NTHR$ exceeds NNP, a warning will be issued in output file $for006$ and $NTHR$ will be reset to 1.

4.1.4 Nodal Data

Format

Prepare NNP lines of data, one for each node, to define the data pertaining to nodes of the structural model in the following form:

X Y Z XRES YRES ZRES RXRES RYRES RZRES FX FY FZ MX MY MZ IDASSCOL

Description

Variable	Field	Notes	Definition
X	1	(1)	X coordinate of node-point.
Y	2	(1)	Y coordinate of node-point.
Z	3	(1)	Z coordinate of node-point.
$XRES$	4	(2)	Restraint to nodal translation in global X direction.
$YRES$	5	(2)	Restraint to nodal translation in global Y direction.
$ZRES$	6	(2)	Restraint to nodal translation in global Z direction.
$RXRES$	7	(3,7)	Restraint to nodal rotation about joint local $\bar{X}$ axis.
$RYRES$	8	(4,7)	Restraint to nodal rotation about joint local $\bar{Y}$ axis.
$RZRES$	9	(4,7)	Restraint to nodal rotation about joint local $\bar{Z}$ axis.
FX	10	(5)	Static nodal force in the global X direction.
FY	11	(5)	Static nodal force in the global Y direction.
FZ	12	(5)	Static nodal force in the global Z direction.
MX	13	(6)	Nodal mass in weight units in the global X direction.
MY	14	(6)	Nodal mass in weight units in the global Y direction.
MZ	15	(6)	Nodal mass in weight units in the global Z direction.
$IDASSCOL$	16	(7)	Element number of the column that is to be associated with the joint under consideration.

Notes

1. The X, Y, and Z coordinates of all the node-points should be specified in the global coordinate system. It is recommended that the global X and Y axes be oriented in the two principal directions of the building and the global Z axis be oriented in the vertically upward direction.

2. If the node is free to translate in the specified direction, this variable should be assigned a value of 1. If its not, then it should be assigned a value of 0. Translational restraints can be specified in only 3 directions, the global X, Y, and Z directions.

3. Rotational restraints need to be specified for the node about the 3 axes of the panel-zone (joint) coordinate system. The panel zone coordinate system starts out to be identical to the element coordinate system corresponding to the associated column of the joint (ACJ). Thus, rotation about the $\bar{X}$ axis corresponds to the twisting DOF of the joint. If the joint is not free to twist, this variable should be assigned a value of 0. If it is free to rotate, then this variable should be assigned a value of 1.

4. Rotational restraints need to be specified for the node about the 3 axes of the panel-zone (joint) coordinate system. The panel zone coordinate system starts out to be identical to the element coordinate system corresponding to the associated column of the joint (ACJ). Thus, rotations about the $\bar{Y}$ and $\bar{Z}$ axes correspond to rotations of the joint about the ACJ major and minor axes respectively. If the joint is not free to rotate, this variable should be assigned a value of 0. If it is free to rotate and if panel zone effects in the direction under consideration are not required to be considered, then this variable should be assigned a value of 1. If panel zone effects are to be considered, then this variable should be assigned a value of 2.

5. These variables define the static forces that are to be applied to the current node in the global X, Y, and Z directions. These static forces can be applied in $NSSTPS$ steps (a fraction of these loads is applied at each step).

6. These variables define the nodal masses in the three global directions, X, Y, and Z. They are to be given in weight units in conjunction with the appropriate acceleration due to gravity, $AGRAV$, that is then used to convert them to masses. These nodal masses are used to compute the dynamic forces on the structure from earthquakes for which the acceleration records are to be provided in input files $for090$, $for091$, and $for092$.

7. This variable defines the identity of the column that is to be associated with the joint at the current node point. The associated column web and flange dimensions along with the depths of beams framing into the joint are used to compute the depth, width and the height of the joint. These dimensions are then used to determine the locations of the attachment points associated with the current node-point at which beams and columns converge. This variable also defines the panel zone coordinate system at the start of the analysis. The panel zone coordinate system starts out as being identical to the associated column element local coordinate system. As the analysis proceeds, the attachment point coordinates are updated and further updates of the panel zone coordinate system are based on these updated attachment point coordinates.

   For gravity columns not connected to any beams, $IDASSCOL$ should be assigned a value of 0. For gravity columns that have gravity beams framing into them, specify a non-zero $IDASSCOL$. In this case, clear length of the column will be taken for stiffness computation.

   Note that in the output file, $for006$, $DOFX$, $DOFY$ & $DOFZ$, refer to the global axes, while $DOFRX$, $DOFRYB$, $DOFRYC$, $DOFRZB$, and $DOFRZC$, refer to the panel zone local axes. If, however, $IDASSCOL$ of a particular node is 0, then there cannot be any panel zones associated with that node. Hence, $DOFRYC$ and $DOFRZC$ are meaningless and the rotations now refer to the global axes, i.e., in this case, the panel zone coordinate system is synonymous with the global coordinate system.

4.1.5 Spring Element Data

Format

Prepare $NSPR$ lines of data, one for each spring, to define the data pertaining to springs in the structural model in the following form:

NODE DOF SPRSTF

Description

Variable	Field	Notes	Definition
$NODE$	1	(1)	Identity of the node-point at which the spring is to be located.
DOF	2	(1)	The degree-of-freedom (1-8) the motion of which is resisted by the spring.
$SPRSTF$	3	(1)	Spring stiffness

Notes

1. Springs can be added to the structural model at any node to resist a translational or rotational degree of freedom (DOF) at that node-point. Multiple springs can be located at each node. Springs can be used to model elastic foundations. They can also be used to retrieve reaction forces and moments at support points. In this case, the spring constant/stiffness ($SPRSTF$) should be specified to be much greater (at least four orders of magnitude greater) than the stiffness of the other elements in the model.

4.1.6 Beam Element Data

Format

Prepare $NPHEL + NFIBEL$ lines of data, one for each beam element, to define the data pertaining to plastic hinge and elastofiber beam elements in the structural model in the following form:

NODE1 NODE2 IBC IDES EE HSH SIGY FAC1 FAC2 ALPHA IELTYPE OLFSRATIO ES SIGU EPSS EPSU ICATFAFRAC ICATFYFRAC ECCMIN[2] ECCMAJ[2]

Description

Variable	Field	Notes	Definition
$NODE1$	1	(-)	Node number at end I of the element.
$NODE2$	2	(-)	Node number at end J of the element.
IBC	3	(1)	Beam or column identifier: 1 for beam, 2 for column, 3 for brace.
$IDES$	4	(2)	Section designation.
EE	5	(3)	Modulus of Elasticity.
HSH	6	(4)	Strain-hardening modulus.
$SIGY$	7	(5)	Yield stress.
$FAC1$	8	(6)	Degree of continuity at end I.
$FAC2$	9	(6)	Degree of continuity at end J.
$ALPHA$	10	(7)	Element orientation angle.
$IELTYPE$	11	(-)	Element type: 0 for plastic-hinge, 1 for elastofiber, 2 for MEF.
$OLFSRATIO$	12	(8)	Ratio of fiber segment length to element total length.
ES	13	(9)	Fiber tangent modulus (slope of stress-strain curve) for an MEF/elastofiber element at onset of strain-hardening.
$SIGU$	14	(10)	Fiber ultimate stress for an MEF/elastofiber element.
$EPSS$	15	(11)	Fiber strain at onset of strain-hardening for an MEF/elastofiber element.
$EPSU$	16	(12)	Fiber ultimate strain for an MEF/elastofiber element.
$ICATFAFRAC$	17	(13)	Category for fiber area reduction.
$ICATFYFRAC$	18	(14)	Category for fiber fracture.
$ECCMIN$	19	(15)	Member initial imperfection in minor direction, along Y' axis. (MEF elements only)
$ECCMAJ$	20	(15)	Member initial imperfection in major direction, along Z' axis. (MEF elements only)

[2]Introduced in Version 2.0

Notes

1. A beam element needs to be identified specifically as a beam, a column or a brace as the element orientation angle, $ALPHA$, is defined differently for beams and columns. For braces, the member orientation angle is defined in an identical fashion as to beams.

2. This variable is the label that identifies the section properties listed in input section properties file, $for009$. It should match with one of the numbers listed in the first column of that file.

3. This variable defines the Young's modulus and is used for both plastic hinge and elastofiber elements.

4. This variable defines the post-yield slope of the moment-rotation relationship for the plastic hinge element at the hinge location as a fraction of $\frac{6EI}{L}$, i.e., the post-yield slope is taken to be $HSH * \frac{6EI}{L}$. Not used for elastofiber/MEF element, input 0.0.

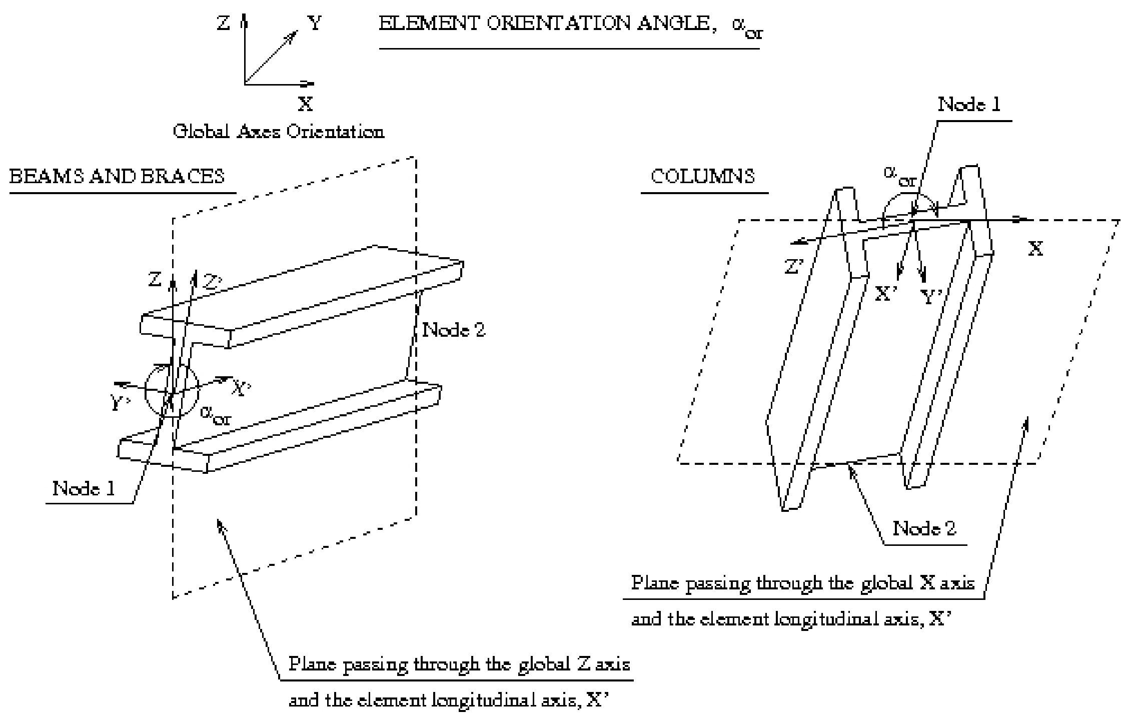
5. This variable defines the material yield stress and is used for both plastic hinge and elastofiber elements.

6. To model pinned-end conditions, the plastic moment capacity, M_p needs to be set to zero. This variable defines the degree of continuity at the end of the element. It should be set to 0.0 if the member end is pinned and 1.0 if the member end is continuous.

7. This variable defines the element orientation angle. It is defined differently for beams and columns as illustrated on Figure 4.1. For columns, α is the angle made by the Z' axis with the global XZ plane, measured using the right-hand-thumb rule with the thumb pointing in the positive direction of the X' axis. For beams, α is the angle made by the Z' axis with a global vertical plane passing through the longitudinal axis of the element, X', measured using the right-hand-thumb rule with the thumb pointed in the positive direction of X' axis. Note that as the analysis proceeds, this angle is updated using the average incremental twist (torsional deformation) in the element at each iteration of each time step.

8. This variable defines the ratio of the length of the end fiber segment (and middle fiber segment in the case of MEF type) to the total clear span of the element for an elastofiber/MEF element. Based on calibration studies on beams in double-curvature, it is recommended that for materials with low ultimate stress to yield stress ratios (less than 1.4) such as A572-Grade 50 steels, end segment lengths of 3% of the clear span be used while for materials with high ultimate stress to yield stress ratios (greater than 1.4) such as A36 steels, end segment lengths of 11% of the clear span be used. For cantilever beams (in single curvature), these values may be doubled. For buckling-sensitive slender columns and braces, a value of 0.02 will yield best results in as far as modeling the elastic and inelastic, critical and post-buckling behavior of the element is concerned (Krishnan 2009). This variable is not used for a plastic hinge element, input 0.0.

9. This variable defines the slope of the axial stress-strain curve for a fiber in an elastofiber/MEF element at the onset of strain hardening (Figure 3.10). A reasonable value for this modulus is 2% of the elastic



BEAMS AND BRACES: α_{or} = Angle made by the Z' axis with a plane passing through the global Z axis and the element longitudinal axis, X' it is measured using the right–hand–thumb rule with the thumb pointing in the positive direction of the X' axis, which runs from node 1 to node 2 of the element

COLUMNS: α_{or} = Angle made by the Z' axis with a plane passing through the global X axis and the element longitudinal axis, X'; it is measured using the right–hand–thumb rule with the thumb pointing in the positive direction of the X' axis, which runs from node 1 to node 2 of the element

Figure 4.1: Beam/Brace/Column Element Orientation Angle

modulus in the case of structural steel. It is not used for a plastic hinge element, input 0.0. Note that the parameters of the fiber axial stress-strain curve must satisfy the following condition, failing which an error will be generated and the program will be terminated: $E_s * \frac{\epsilon_u - \epsilon_{sh}}{\sigma_u - \sigma_y} \leq 3.0$

10. This variable defines the ultimate axial stress for a fiber in an elastofiber/MEF element (Figure 3.10). It is not used for a plastic hinge element, input 0.0.

11. This variable defines the strain in the fiber of an elastofiber/MEF element at the onset of strain-hardening (Figure 3.10). A reasonable value for this strain is 1.2% in the case of structural steel. It is not used for a plastic hinge element, input 0.0.

12. This variable defines the ultimate strain in the fiber of an elastofiber/MEF element (Figure 3.10). A reasonable value for this strain is 16.0% in the case of structural steel. It is not used for a plastic hinge element, input 0.0.

13. To model partial penetration welds or other forms of connections that lead to reduced moment capacities than dictated by gross properties of flanges and webs, it may be necessary to reduce the area of fibers in the flanges and/or webs. This can be done by classifying all the elastofiber/MEF elements into fiber area reduction categories, with each category representing a certain reduction pattern in the 20 fibers of each segment of an elastofiber/MEF element. This variable defines the fiber area reduction category that the current elastofiber/MEF element belongs to. If a value of 0 is specified, no reduction is applied to the fiber area. It is not used for a plastic hinge element, input 0.

14. In some moment connections, premature fracture could lead to failure in beam elements prior to the rupture stress being reached. To model this mode of failure, it may be necessary to specify a fracture strain for each fiber based on an assumed probability distribution. This can be done by classifying all the elastofiber/MEF elements into different fracture strain distribution categories, with each category representing a certain probability distribution of fracture strains for the fibers of each segment of an elastofiber/MEF element. This variable defines the fracture strain distribution category that the current elastofiber/MEF element belongs to. By default, a very large fracture strain (far greater than the rupture strain) is assumed for all fibers, unless they are assigned an $ICATFYFRAC$ greater than zero. Assign a value of 0 for $ICATFYFRAC$ if fracture is not to be considered. This variable is not used for a plastic hinge element, input 0.

15. The modeling of axial response of braces and slender columns is very sensitive to geometric imperfections in these members. $ECCMAJ$ and $ECCMIN$ allow for the modeling of these imperfections. The locations of the two interior nodes defining the middle segment of a 5-segment modified elastofiber element are displaced by an amount of $ECCMAJ$*OL in the major direction of the cross-section, and an amount of $ECCMIN$*OL in the minor direction of the cross-section, where OL is the original clear span of the member. For a perfectly concentric axially loaded pin-ended member, a value of 0.000005 needs to be specified for these variables to allow for the initiation of buckling. It is recommended that a default value of 0.000005 be used for these variables if buckling needs to

be simulated. These variables are used only for 5-segment MEF elements. Specify zero for other elements.

4.1.7 Categories for Elastofiber/MEF Element Fiber Area Reduction

Format

Prepare required lines of data, **followed by a line of zeros**, to define the fiber area reduction categories for elastofiber/MEF elements in the structural model in the following form:

ICAT IDSEG IDFIB(20) FAFRAC

Description

Variable	Field	Notes	Definition
$ICAT$	1	(1)	Fiber area reduction category number.
$IDSEG$	2	(1,2)	Fiber segment 1 or 2.
$IDFIB(20)$	3-22	(1,3)	Fiber numbers.
$FAFRAC$	23	(1,4)	Fraction of fiber area to be used in the analysis (0.0-1.0).

Notes

1. This variable identifies the category for which data is provided on this line. Note that each category could have multiple lines of data. For example, the first 6 lines of this data set could belong to category 1, with the first line being applicable to segment 1 top flange fibers, the second line being applicable to segment 1 bottom flange fibers, the third line being applicable to segment 1 web fibers, with the fourth, fifth and sixth lines being applicable to the corresponding data for segment 2. Many such combinations are possible for each category. **This is why this data block needs to be followed by a line of zeros**.

2. This variable defines the segment number of the elastofiber element (1 or 2) or the modified elastofiber element (1 or 2 or 3) for which this line of data is applicable.

3. Provide 20 numbers corresponding to the fibers of the segment, $IDSEG$, for which the fiber area reduction data provided on this line is applicable. If $FAFRAC$ data on this line is applicable to fewer than 20 fibers, then pad the fiber numbers with zeros such that 20 numbers are provided covering Fields 3-22 of the data line.

4. This variable defines the fraction of fiber area to be used for the fibers specified in $IDFIB(20)$ of the segment, $IDSEG$, for elastofiber/MEF elements belonging to the current category, $ICAT$.

4.1.8 Seed for Random Number Generation for Elastofiber/MEF Element Fiber Fracture Strain Computation

Format

Prepare one line of data to define the seed for random number generation to be used for picking a probabilistic fracture stress for fibers of an elastofiber/MEF element in the following form:

ISEED

Description

Variable	Field	Notes	Definition
$ISEED$	1	(1)	Seed to generate random numbers.

Notes

1. This variable is used as a seed to generate a random number for each line of data of the fiber area reduction category of each elastofiber/MEF element which is then used to pick one of the 10 fracture strains listed on that line of that category to be used for all the fibers listed on that line of that category. $ISEED$ needs to be between -1 and 1000; if $ISEED$ = -1 then the random number generator is seeded using the milliseconds on the system clock as seed for srand(); the resulting seed is output into the input file so that results can be reproduced by changing $ISEED$ to this value and rerunning; if $0 \leq ISEED \leq 1000$ then the random number generator is seeded using $ISEED$ as seed for srand(). Random numbers are then generated sequentially using rand(0).

4.1.9 Categories for Elastofiber/MEF Element Fiber Fracture Strain Probability Distribution

Format

Prepare required lines of data, **followed by a line of zeros**, to define the fiber fracture probability distribution categories for elastofiber/MEF elements in the structural model in the following form:

ICAT IDSEG IDFIB(20) FYFRAC(10)

Description

Variable	Field	Notes	Definition
$ICAT$	1	(1)	Fiber fracture stress probability distribution category number.
$IDSEG$	2	(1,2)	Fiber segment 1 or 2.
$IDFIB(20)$	3-22	(1,3)	Fiber numbers.
$FYFRAC(10)$	23-32	(1,4)	Fracture strain probability distribution.

Notes

1. This variable identifies the category for which data is provided on this line. Note that each category could have multiple lines of data. For example, the first 6 lines of this data set could belong to category 1, with the first line being applicable to segment 1 top flange fibers, the second line being applicable to segment 1 bottom flange fibers, the third line being applicable to segment 1 web fibers, with the fourth, fifth and sixth lines being applicable to the corresponding data for segment 2. Many such combinations are possible for each category. **This is why this data block needs to be followed by a line of zeros**.

2. This variable defines the segment number of the elastofiber element (1 or 2) or the modified elastofiber element (1 or 2 or 3) for which this line of data is applicable.

3. Provide 20 numbers corresponding to the fibers of the segment, $IDSEG$, for which the fiber fracture strain distribution data provided on this line is applicable. If $FYFRAC$ data on this line is applicable to fewer than 20 fibers, then pad the fiber numbers with zeros such that 20 numbers are provided covering Fields 3-22 of the data line.

4. Provide 10 numbers specifying the fracture strain distribution for the fibers specified in $IDFIB(20)$ of the segment, $IDSEG$, for elastofiber/MEF elements belonging to the current category, $ICAT$. Note that each of these numbers are specified as a fraction of the yield strain, i.e., $\epsilon_{frac} = FYFRAC * \epsilon_y$. For each elastofiber/MEF element with $ICATFYFRAC$ that is identical to the current category, $ICAT$, a random number $1 \leq IRAND \leq 10$ is generated. Each fiber in $IDFIB(20)$ of segment $IDSEG$ of that element is then assigned a fracture strain of $\epsilon_{frac} = FYFRAC(IRAND) * \epsilon_y$. For example, let us say that there are five categories of fracture strain probability distributions. Let us

say that for the fifth category ($ICAT = 5$), the fracture strain distribution for all segment 1 bottom flange fibers is defined as follows: $FYFRAC = [0.9, 0.9, 2.0, 2.0, 2.0, 5.0, 5.0, 5.0, 40.0, 40.0]$. Now consider an element with $ICATFYFRAC = ICAT = 5$. Now let's assume that the generated random number $IRAND = 4$. Then for the bottom flange fibers of segment 1 of this element, the fracture strain would be taken to be $FYFRAC(IRAND) * \epsilon_y = FYFRAC(4) * \epsilon_y = 2.0\epsilon_y$. Note that this fracture strain distribution implies that there is 20% chance that the fracture strain would be 0.9 times the yield strain, 30% chance that it would be 2.0 times the yield strain, 30% chance that it would be 5.0 times the yield strain, and 20% chance that it would be 5.0 times the yield strain.

4.1.10 Panel Zone Element Data

Format

Prepare $NPZ0 + NPZ1$ lines of data, one for each panel zone, to define the data pertaining to panel zone elements in the structural model in the following form:

NODE NPZZORY IPZTYPE G HSH TAUY THK DTHK HPZ

Description

Variable	Field	Notes	Definition
$NODE$	1	(1)	Node number.
$NPZZORY$	2	(2)	Panel zone orientation: 1 for $X'Z'$ plane and 2 for $X'Y'$ plane.
$IPZTYPE$	3	(3)	Panel zone type: 0 for bilinear and 1 for linear-quadratic.
G	4	(4)	Shear modulus.
HSH	5	(5)	Strain hardening modulus.
$TAUY$	6	(6)	Shear yield stress.
THK	7	(7)	Panel zone thickness.
$DTHK$	8	(8)	Doubler plate thickness.
HPZ	9	(9)	Panel zone height.

Notes

1. Each node can have up to 2 panel zones in two orthogonal directions. This variable defines the node number at which a panel zone is to be located. Note that the nodal restraints specified for this node in the nodal data block must suitably reflect the existence of a panel zone (i.e., with either the $DOFRY$ variable or the $DOFRZ$ variable being assigned a value of 2).

2. This variable defines the orientation of the panel zone whether it is in the $X'Z'$ plane of the associated column of this joint (ACJ), corresponding to the major axis bending of the ACJ or whether it is in the $X'Y'$ plane of the associated column of this joint (ACJ), corresponding to the minor axis bending of the ACJ. Assign a value of 1 for panel zone in the $X'Z'$ plane of ACJ and a value of 2 for panel zone in the $X'Y'$ plane of ACJ.

3. This variable defines the shear stress-strain law for the panel zone material. The stress-strain relation could be bilinear (Figure 3.2) in which case, this variable should be assigned a value of 0, or linear-quadratic (Figure 3.3) in which case, this variable should be assigned a value of 1.

4. This variable defines the elastic shear modulus of the panel zone material. For steel, the shear modulus can be taken to be $G = \frac{E}{2.5}$.

5. This variable defines the post-yield slope of the shear stress-strain relationship for a bilinear panel zone as a fraction of G, i.e., the post-yield slope is taken to be $HSH * G$. Not used for elastofiber element, input 0.0.

6. This variable defines the shear stress in the panel zone at which yield occurs. Per the Von-Mises yield criterion, the shear yield stress can be taken to be $\tau_y = \frac{\sigma_y}{\sqrt{3}}$, where σ_y is the tensile yield stress of the material.

7. This variable defines the thickness of the panel zone to be used for the analysis. Assign a negative number to this variable if the web/flange thickness of the ACJ is to be used. If NPZZORY is 1, then the sum of the thicknesses of all the webs of the ACJ will be used and if NPZZORY is 2, then the sum of the thicknesses of all the flanges of the ACJ will be used.

8. This variable defines the thickness of any doubler plates that have to be used to reinforce the joint. The program can compute the doubler plate thickness if the connecting beam is bending about its major axis using equation (13-1), Section 2213.7.2.1 of the Uniform Building Code, UBC97 (ICBO 1997). Specify a negative value for $DTHK$ if the doubler plate thickness is to be internally calculated by the program.

9. This variable defines the height of the panel zone to be used in the analysis. Assign a value of the negative of the beam ID if the panel zone height needs to be assigned the depth of a certain beam. Thus, if this variable was assigned an integer value of -500, then, the height of the panel zone would be taken to be equal to the depth of beam element, 500.

4.1.11 Diaphragm Element Control Data

Format

Prepare one line of data, giving the control data for elastic diaphragm elements in the structural model in the following form:

NUMPLSTREL NUMPLSTRMAT NINTPTPLSTR NPLSTRSS

Description

Variable	Field	Notes	Definition
$NUMPLSTREL$	1	(-)	Number of diaphragm elements.
$NUMPLSTRMAT$	2	(-)	Number of diaphragm element material types.
$NINTPTPLSTR$	3	(1)	Number of integration points for Gauss quadrature.
$NPLSTRSS$	4	(2)	Number of diaphragm elements for which principal stress-strain history is required.

Notes

1. This variable defines the number of integration points to be used for Gauss quadrature of a diaphragm element. If a 1-point Gauss quadrature is to be used, assign a value of 1 to this variable. If a 2x2 Gauss quadrature with 4 integration points is to be used, assign a value of 2 to this variable.

2. This variable defines the number of diaphragm elements for which principal stress-strain history is required. Output is presented in file, $PLSTRSS$, and includes the two principal stresses, the angle of the principal plane and the maximum shear stress in the element at each time step of the analysis.

4.1.12 Diaphragm Element Material Data

Format

Prepare $NUMPLSTRMAT$ lines of data, one for each diaphragm element material type, to define the data pertaining to diaphragm element materials in the structural model in the following form:

E POIS

Description

Variable	Field	Notes	Definition
E	1	(1)	Elastic modulus.
$POIS$	2	(1)	Poisson's ratio.

Notes

1. For each of the $NUMPLSTRMAT$ materials, provide the elastic modulus and the Poisson's ratio. For concrete slabs, the elastic modulus can be taken to be $57000\sqrt{f_c'}$ while the Poisson's ratio can be taken to be 0.30.

4.1.13 Diaphragm Element Connectivity Data

Format

Prepare required number of lines of data, to generate the node connectivity of diaphragm elements and assign their material properties in the following form. **End with a line of zeros**.

N NE NG MAT THK N1 N2 N3 N4 LMG

Description

Variable	Field	Notes	Definition
N	1	(1,4)	First element in the series to be generated.
NE	2	(1,4)	Last element in the series to be generated.
NG	3	(1,4)	Increment in diaphragm element number.
MAT	4	(2)	Material number.
THK	5	(3)	Diaphragm element thickness.
$N1$	6	(1,4)	Node 1 of first element in the series to be generated.
$N2$	7	(1,4)	Node 2 of first element in the series to be generated.
$N3$	8	(1,4)	Node 3 of first element in the series to be generated.
$N4$	9	(1,4)	Node 4 of first element in the series to be generated.
LMG	10	(5)	Increment to be applied to the node numbers $N1$-$N4$ for the next element in the series to be generated.

Notes

1. For diaphragm elements, N, $N + NG$, $N + NG * 2$,...., $NE - NG$, and NE, the element material set is set to MAT and the thickness is set to THK.

2. This variable defines the ID of the material property set that needs to be assigned to elements, N, $N + NG$, $N + NG * 2$,...., $NE - NG$, and NE. The material property sets are defined in the Plane Stress Element Material Data block.

3. This variable defines the thickness of diaphragm elements, N, $N+NG$, $N+NG*2$,...., $NE-NG$, and NE.

4. The nodal connectivity for elements, N, $N + NG$, $N + NG * 2$,...., $NE - NG$, and NE, is taken to be $[N1, N2, N3, N4]$, $[N1+LMG, N2+LMG, N3+LMG, N4+LMG]$, $[N1+LMG*2, N2+LMG*2, N3+LMG*2, N4+LMG*2]$,...., $[N1+LMG*(\frac{NE-N}{NG}-1), N2+LMG*(\frac{NE-N}{NG}-1), N3+LMG*(\frac{NE-N}{NG}-1), N4+LMG*(\frac{NE-N}{NG}-1)]$, and $[N1+LMG*\frac{NE-N}{NG}, N2+LMG*\frac{NE-N}{NG}, N3 + LMG * \frac{NE-N}{NG}, N4 + LMG * \frac{NE-N}{NG}]$ respectively. Note that $N1$, $N2$, $N3$, and $N4$ need to be defined in a cyclic fashion, otherwise, a negative Jacobian will result in abrupt termination of the program.

4.1.14 Diaphragm Element Output Control Data

Format

Prepare required number of lines of data, to specify the plane stress elements for which principal stress-strain histories are required, in the following form:

N1 N2

Description

Variable	Field	Notes	Definition
$N1$	1	(1)	First diaphragm element in the series for which principal stress-strain histories are required.
$N2$	2	(1)	Last diaphragm element in the series for which principal stress-strain histories are required.

Notes

1. These variables define the series of diaphragm elements for which time-histories of principal stresses are required. Provide as many lines as required. The total number of elements listed here must add up to the input variable, $NPLSTRSS$, specified in the Plane Stress Element Control Data block.

4.1.15 Time History Output Control Data

Format

Prepare $NRTH$ lines of data, specifying the nodal and element response quantities for which time-histories are required, in the following form[3]:

NODE RESPID ELEM RESPID PZ RESPID ELEM SEG FIB RESPID

Description

Variable	Field	Notes	Definition
$NODE$	1	(1)	Node number.
$RESPID$	2	(2)	Nodal response quantity.
$ELEM$	3	(3)	Beam element number.
$RESPID$	4	(4)	Element response quantity.
$NODE$	5	(5)	Node number.
$RESPID$	6	(6)	Panel zone response quantity.
$ELEM$	7	(7)	Elastofiber/MEF beam element number.
SEG	8	(8)	Segment number.
FIB	9	(9)	Fiber number.
$RESPID$	10	(10)	Fiber response quantity.

Notes

1. This variable defines the node number for which the time-history of a nodal response quantity, $RESPID$, is required to be output in files $for006$ and RTH. Assign a value of zero if a nodal response quantity is not required.

[3]Changes have been made in Versions 1.1 and 2.0

2. This variable defines the nodal response quantity of $NODE$, the time-history of which is required to be output in files $for006$ and RTH. The various nodal response quantities and the corresponding values for $RESPID$ are listed below. Assign a value of zero if a nodal response quantity is not required.

Input	Response Quantity
0	If element or panel zone or fiber response quantity is required.
1	U (Global X direction) DOF displacement.
2	V (Global Y direction) DOF displacement.
3	W (Global Z direction) DOF displacement.
4	$\theta_{\bar{X}}$ (Panel zone coordinate system $\bar{X}$ direction) DOF rotation.
5	$\theta_{\bar{Y}}$ (Panel zone coordinate system $\bar{Y}$ direction) DOF rotation.
7	$\theta_{\bar{Z}}$ (Panel zone coordinate system $\bar{Z}$ direction) DOF rotation.
11	X-direction elastic spring internal force.
12	Y-direction elastic spring internal force.
13	Z-direction elastic spring internal force.
21	U (global coordinate system X direction) DOF velocity.
22	V (global coordinate system Y direction) DOF velocity.
23	W (global coordinate system Z direction) DOF velocity.
31	U (global coordinate system X direction) DOF acceleration.
32	V (global coordinate system Y direction) DOF acceleration.
33	W (global coordinate system Z direction) DOF acceleration.

3. This variable defines the beam element number for which the time-history of an element response quantity, $RESPID$, is required to be output in files $for006$ and RTH. Assign a value of zero if a beam element response quantity is not required.

4. This variable defines the beam element response quantity of $ELEM$, the time-history of which is required to be output in files $for006$ and RTH. The various beam element response quantities and the corresponding values for $RESPID$ are listed below. Assign a value of zero if a beam element response quantity is not required.

Input	Response Quantity
0	If nodal or panel zone or fiber response quantity is required.
1	Moment, $M_{X'}$, about element local X' axis (Torsion).
2	Moment, $M^1_{Y'}$, at node 1 about element local Y' axis (Major axis bending).
3	Moment, $M^2_{Y'}$, at node 2 about element local Y' axis (Major axis bending).
4	Moment, $M^1_{Z'}$, at node 1 about element local Z' axis (Minor axis bending).
5	Moment, $M^2_{Z'}$, at node 2 about element local Z' axis (Minor axis bending).
6	Kink angle, $\kappa^1_{Y'}$, at node 1 about element local Y' axis (Major axis bending).
7	Kink angle, $\kappa^2_{Y'}$, at node 2 about element local Y' axis (Major axis bending).
8	Kink angle, $\kappa^1_{Z'}$, at node 1 about element local Z' axis (Minor axis bending).
9	Kink angle, $\kappa^2_{Z'}$, at node 2 about element local Z' axis (Minor axis bending).
10	Axial force, P, in element.
11	Axial strain, ϵ, in element.
12	Updated length, L, of element.
13	CASEY for element.
14	CASEZ for element.
20	Axial force in middle segment of 5-segment modified elastofiber element.
21	Minor direction lateral displacement of middle segment of MEF element (to track buckling).
22	Major direction lateral displacement of middle segment of MEF element (to track buckling).
23	SRSS of 21 and 22.
24	Shear, $Q^1_{Y'}$, at node 1 in the element local Y' direction.
25	Shear, $Q^2_{Y'}$, at node 2 in the element local Y' direction.
26	Shear, $Q^1_{Z'}$, at node 1 in the element local Z' direction.
27	Shear, $Q^2_{Z'}$, at node 2 in the element local Z' direction.

5. This variable defines the node number corresponding to the panel zone element for which the time-history of a panel zone response quantity, $RESPID$, is required to be output in files $for006$ and RTH. Assign a value of zero if a panel zone element response quantity is not required.

6. This variable defines the response quantity of the panel zone element at $NODE$, the time-history of which is required to be output in files $for006$ and RTH. The various panel zone element response quantities and the corresponding values for $RESPID$ are listed below. Assign a value of zero if a panel zone element response quantity is not required.

Input	Response Quantity
0	If nodal or beam element or fiber response quantity is required.
1	Moment, $M_{\bar{Y}}$, about panel zone element local $\bar{Y}$ axis in the ACJ $X'Z'$ panel zone (corresponding to the major axis bending of the ACJ).
2	Shear strain, $\gamma_{\bar{Y}}$, about panel zone element local $\bar{Y}$ axis in the ACJ $X'Z'$ panel zone (corresponding to the major axis bending of the ACJ).
3	Moment, $M_{\bar{Z}}$, about panel zone element local $\bar{Z}$ axis in the ACJ $X'Y'$ panel zone (corresponding to the minor axis bending of the ACJ).
4	Shear strain, $\gamma_{\bar{Z}}$, about panel zone element local $\bar{Z}$ axis in the ACJ $X'Y'$ panel zone (corresponding to the minor axis bending of the ACJ).

7. This variable defines the elastofiber/MEF beam element number for which the time-history of a fiber response quantity, $RESPID$, is required to be output in files $for006$ and RTH. Assign a value of zero if an elastofiber/MEF beam element fiber response quantity is not required.

8. This variable defines the fiber segment ID (1 or 2 for elastofiber element, and 1, 2, or 3 for modified elastofiber element) that contains the fiber for which a fiber response quantity, $RESPID$, is required to be output in files $for006$ and RTH. Assign a value of zero if an elastofiber/MEF beam element fiber response quantity is not required.

9. This variable defines the fiber ID (1-20, Figures 3.8 and 3.12) for which a response quantity, $RESPID$, is required to be output in files $for006$ and RTH. Assign a value of zero if an elastofiber/MEF beam element fiber response quantity is not required.

10. This variable defines the fiber response that is required to be output in files $for006$ and RTH. Assign a value of zero if an elastofiber/MEF beam element fiber response quantity is not required. Assign a value of 1 if fiber stress history is required or a value of 2 if strain history is required.

4.1.16 Element Group Resultant Output Control Data

Format

Prepare one line of data, specifying the number of elements in each of the $NGRP$ groups, in the following form[4]:

GRP-1-NELM GRP-2-NELM ..... GRP-$NGRP$-NELM

followed by $NGRP$*$(GRP-1-NELM+GRP-2-NELM+.....+GRP-NGRP-NELM)$ lines, one for each element of group 1 followed by group 2, and so on until group $NGRP$:

IDELM NODEID

Description

Variable	Field	Notes	Definition
$IDELM$	1	(1)	Element number .
$NODEID$	2	(1)	1 for forces at the left end of element $IDELM$ or 2 for forces at the right end of element $IDELM$.

Notes

1. This section allows the user to specify groups of elements for which combined forces are to be written out at each time step of the analysis. For each element in a group, the internal forces are transformed to the global coordinate system, and forces at the left or right end as specified by the user are added to the forces in all the other elements of the group, and the resultant forces are written out in file, $ELMGRPRES$.

[4] Introduced in Version 2.0

4.1.17 Ground Acceleration Scaling Factors

Format

Prepare one line of data, specifying the ground acceleration scaling factors, in the following form:

GAMULTX GAMULTY GAMULTZ

Description

Variable	Field	Notes	Definition
$GAMULTX$	1	(1)	Scaling factor for X-component of ground acceleration.
$GAMULTY$	2	(1)	Scaling factor for Y-component of ground acceleration.
$GAMULTZ$	3	(1)	Scaling factor for Z-component of ground acceleration.

Notes

1. These variables define the scaling factors to be applied to the ground motion acceleration records provided in input files $for090$, $for091$, and $for092$. They can be used to convert the acceleration records from one system of units to another. Of course, they can be used for direct scaling of records.

4.1.18 Deformed Shape Output Control Data

Format

Prepare one line of data, specifying the time interval at which deformed shape nodal coordinates are required, in the following form:

DTOUT

Description

Variable	Field	Notes	Definition
$DTOUT$	1	(1)	Time interval at which deformed shape nodal coordinates are required.

Notes

1. This variable defines the time interval at which the nodal coordinates of the structure are required as the structure deforms during time-history analysis. Deformed shape nodal coordinate data may require large amounts of storage space and this variable can be used effectively to keep the required storage space in check. Normally, data written at 0.10 seconds should be sufficient to create realistic animations of the structure as it vibrates during, say, an earthquake.

4.1.19 Yield Stress Units

Format

Prepare one line of data, specifying the type of units used for the yield stress of beams, columns, and panel zones, in the following form[5]:

UNITS

Description

Variable	Field	Notes	Definition
$UNITS$	1	(1)	Units in which yield stress, ϵ_y, has been specified. Enter 1 for ksi or 2 for kg/m^2.

Notes

1. This variable identifies the units in which the user has defined the yield stresses of beams, columns, and panel zones. It is used for comparing the plastic rotations in the beam and column sections and the panel zones against FEMA-356 (FEMA 2000) acceptance criteria for various performance levels. Each beam and column end and panel zone is assigned one of three performance levels, Immediate Occupancy (IO), Life-Safe (LS), and Collapse Prevention (CP). If the plastic rotations exceed the these limits then the connection is labeled CO (collapsed).

[5] Introduced in Version 1.1

4.1.20 Peak Interstory Drift Output Control Data

Format

Prepare one line of data, specifying the number of X direction peak interstory drifts and the number of Y direction peak interstory drifts that are required to be output, in the following form[6]:

NDRIFTX NDRIFTY

Description

Variable	Field	Notes	Definition
$NDRIFTX$	1	(-)	Number of stories at which X direction peak interstory drift ratios are to be computed.
$NDRIFTY$	2	(-)	Number of stories at which Y direction peak interstory drift ratios are to be computed.

Format

Then prepare NDRIFTX+NDRIFTY lines of data, specifying the $RESPID$ of node 1 (located at the bottom of the story) displacement, the $RESPID$ of node 2 (located at the top of the story) displacement, and the interstory height, in the following form:

RTH1 RTH2 STORYHT

Description

Variable	Field	Notes	Definition
$RTH1$	1	(1)	ID of the time-history that corresponds to the displacement of the bottom of the story. This has to be one of the $NRTH$ time-histories for which output is to be written according to Section 4.1.15.
$RTH2$	1	(1)	ID of the time-history that corresponds to the displacement of the top of the story. This has to be one of the $NRTH$ time-histories for which output is to be written according to Section 4.1.15.
$STORYHT$	3	(1)	Story height.

Notes

1. Consider a 5-story building with four columns (and hence 4 nodes on each floor) and an interstory height of 4.0m. Suppose we number the nodes at the first floor as 1–4, at the second floor as 5–8 and so on. If we are interested in the interstory drifts at one corner of the building in the X and Y

[6] Introduced in Version 1.1

directions for each story, then $NRTH = 10$ in Section 4.1.15 of the input data, with the first five being the X displacements of nodes 5, 9, 13, 17, and 21, and the next five being the corresponding Y displacements (node 1–4 are assumed fixed). For drift output: $NDRIFTX = 5$, $NDRIFTY = 5$, $RTH1 = [1, 2, 3, 4, 5, 6, 7, 8, 9, 10]$ and $RTH2 = [1, 1, 2, 3, 4, 6, 6, 7, 8, 9]$,
and $STORYHT = [4.0, 4.0, 4.0, 4.0, 4.0, 4.0, 4.0, 4.0, 4.0, 4.0]$. The interstory drift will be computed as $\frac{RTH1-RTH2}{STORYHT}$ when $RTH1 \neq RTH2$, or $\frac{RTH1}{STORYHT}$ when $RTH1 = RTH2$. The input should be formatted as follows:

5 5

1 1 4.0

2 1 4.0

3 2 4.0

4 3 4.0

5 4 4.0

6 6 4.0

7 6 4.0

8 7 4.0

9 8 4.0

10 9 4.0

4.2 Input File for009

File $for009$ is an input data file listing beam element section properties. Beam sections are required to be identified by a positive or a negative integer label. Positive integer labels are reserved for wide-flanged I-sections, whereas, negative integer labels are reserved for box-sections. At the present time, beam cross-sections are limited to wide-flanged and box sections.

4.2.1 Labeling

Prepare one line labeling each of the columns in the data file. This line is treated as a comment line and is not read by the program. It has been inserted for user convenience in working with the data in this section property file.

4.2.2 Beam Section Data

Format

Prepare up to a maximum of 10000 lines of data to define the beam section properties to be used in the structural model in the following form:

IDES AREA SMI WMI SSM WSM SZ WZ DEPTH WEBTHK FLGWDTH FLGTHK TORJ

Description

Variable	Field	Note	Definition
$IDES$	1	(1)	Section designation (label).
$AREA$	2	(-)	Area of the cross-section.
SMI	3	(-)	Moment of inertia about the strong (major) axis of the section.
WMI	4	(-)	Moment of inertia about the weak (minor) axis of the section.
SSM	5	(-)	Section modulus about the strong (major) axis of the section.
WSM	6	(-)	Section modulus about the weak (minor) axis of the section.
SZ	7	(-)	Plastic section modulus about the strong (major) axis of the section.
WZ	8	(-)	Plastic section modulus about the weak (minor) axis of the section.
$DEPTH$	9	(-)	Section depth.
$WEBTHK$	10	(-)	Web thickness.
$FLGWDTH$	11	(-)	Flange width.
$FLGTHK$	12	(-)	Flange thickness.
$TORJ$	13	(-)	Torsional constant.

Notes

1. $IDES$ should be an integer. Assign a positive integer for wide-flanged I-sections and a negative integer for box sections.

4.3 Input Files for090, for091 and for092

Acceleration histories in the global X, Y, and Z directions should be provided in input files $for090$, $for091$, and $for092$ respectively.

4.3.1 Heading

Prepare one line providing the station information, units etc. for the record provided in the data file. This line is treated as a comment line and is not read by the program. It has been inserted for user convenience in identifying the recorded data.

4.3.2 Acceleration Data Points

Format

Prepare required number of lines of data to define at least $NDSTPS$ data points on the acceleration records. Separate each data point from the other with a blank space. Any number of data points can be provided on each line. The total number of data points should be greater than or equal to the input variable, $NDSTPS$, specified in the Main Control Data block.

Chapter 5   Program Output

Ten output files are generated from a FRAME3D analysis. These are listed in Table 5.1.

Table 5.1: Output Files from FRAME3D Analysis

File Name	Description
$for006$	Comprehensive output file.
RTH	Node and element response time-histories.
$COORD$	Undeformed coordinates of the structure.
$DEFCOORD$	Deformed coordinates of the structure.
$BEAM$	Beam element connectivity.
$PLSTR$	Diaphragm element connectivity.
$BMPLROT$	Beam element plastic rotations.
$PZPLROT$	Panel zone element plastic rotations.
$PLSTRSS$	Diaphragm element principal stress time-histories.
$PLSTRSMAX$	Diaphragm element peak principal stresses.
$FRAC$	List of elastofiber/MEF element fiber fractures.
$FRACSUM$	List of elastofiber/MEF element segments with fracture in at least one fiber.
$FRACTOT$	Total number of segments of elastofiber/MEF elements with fracture in at least one fiber.
RUP	List of elastofiber/MEF element fiber ruptures.
$FAIL$	List of elastofiber/MEF element segments that completely fail.
$XDRFT$	Peak interstory drifts in the X direction.
$YDRFT$	Peak interstory drifts in the Y direction.
$AVGPKDRFT$	Maximum of the average of the peak interstory drifts in the X and Y directions.
$PKDRFT$	The absolute peak interstory drift in the model (X or Y direction).
$XDRFTRTH$	X drift time histories.
$YDRFTRTH$	Y drift time histories.
$FEMA356$	Summary of structure performance based on beam, column, and panel zone plastic rotations against FEMA356 acceptance criteria.
$PERF$	Classification of building performance based on FEMA356 acceptance criteria.
$EIGEN$	Estimated Eigen values and modal periods.
$MODES$	Estimated Eigen vectors (mode shapes).
$BUCKLING$	Deformed coordinates of the middle interior nodes of 5-segment MEF elements for buckling tracking and visualization.
$ELMGRPRES$	Element group resultant forces in the global coordinate frame of reference.
PMM	Discretized, normalized PMM interaction surface for plastic hinge elements.
DAM_IND_EF	Damage index of end segments of elastofiber elements.
$DAM\_IND\_EF5\_END$	Damage index of end segments of MEF elements.
$DAM\_IND\_EF5\_MID$	Damage index of middle segment of MEF elements.
DAM_IND_PZ1	Damage index of type 1 panel zones (with linear-quadratic shear stress-strain backbone relation).

The following sections describe the output provided in each file in greater detail.

5.1 File for006

$for006$ is the main output file that is generated from a FRAME3D analysis. Output includes echoing of the input as well as user required responses in addition to other critical analysis information generated by the program such as global degree of freedom numbers etc. The various sections in the output file are listed in Table 5.2. Notes are provided where new variables are encountered as well as where program generated information is printed out. For all other variables and their definitions, refer to Section 4.1.

Table 5.2: File for006: List of Output Sections

S.No.	Output Section	Notes
1.	Heading	(-)
2.	Main Control Data	(-)
3.	Integration and Iteration Data	(-)
4.	Nodal Data	(-)
5.	Degree of Freedom Numbers	(1)
6.	Spring Element Data	(-)
7.	Joint Dimensions	(2)
8.	Categories for Elastofiber/MEF Element Fiber Area Reduction	(-)
9.	Seed for Random Number Generation for Elastofiber/MEF Element Fiber Fracture Strain Computation	(-)
10.	Categories for Elastofiber/MEF Element Fiber Fracture Strain Probability Distribution	(-)
11.	Beam Element Data	(3)
12.	Elastofiber/MEF Element Fiber Fracture Strain	(4)
13.	Panel Zone Element Data	(5)
14.	Diaphragm Element Control Data	(-)
15.	Diaphragm Element Material Data	(-)
16.	Diaphragm Element Connectivity Data	(-)
17.	Diaphragm Element Output Control Data	(-)
18.	Bandwidth Information	(6)
19.	Time History Output Control Data	(-)
20.	Static Solution	(7)
21.	Static Element Responses	(8)
22.	Time History Data	(9)
23.	Dynamic Solution	(10)
24.	Minimum Static Plus Dynamic Element Responses	(11,8)
25.	Maximum Static Plus Dynamic Element Responses	(11,8)
26.	Panel Zone Inelastic Rotations and Performance Levels	(12)
27.	Beam End Inelastic Rotations and Performance Levels	(13)
28.	Summary of Structure Performance	(14)
29.	Principal Stresses and Maximum Shear Stresses in Diaphragm Elements	(15)
30.	Building Damage Indices (Elastofiber Elements, MEF Elements, and Type 1 Panel Zone Elements)	(16)
31.	Peak Interstory Drifts and Structure Performance Level Based on Drift	(17)
32.	Elastofiber/MEF Element Fracture Details	(18)
33.	Total execution time	(-)

Notes

1. In this section of the output, the total number of degrees of freedom, $NDOF$, in the structural model as computed by the program is printed. In addition, the global degrees of freedom numbers from 1 to $NDOF$ corresponding to the 8 possible degrees of freedom at each node are listed. A zero against a nodal degree of freedom indicates that that degree of freedom is restrained. An identical degree of freedom number for $DOFRYB$ and $DOFRYC$ or $DOFRZB$ and $DOFRZC$ indicates a rigid panel zone at that node in the corresponding direction, whereas distinct numbers for these indicates the presence of a panel zone with bilinear or linear-quadratic shear stress-strain behavior.

2. In this section, the height, depth and width of the joint at each node are printed. The height of the joint is computed as the maximum depth of all the beams framing into that joint. The depth and width of the joint are taken to be the same as those of the associated column (ACJ). These dimensions are then used to determine the location of the 6 attachment points associated with the joint at which beams and columns connect. Zero dimensions indicate a free or restrained end, i.e., no beam-column joint exists at this node.

3. In addition to reflecting the input data provided by the user, this section provides some more data that is internally generated by the program. $N1ATTACH$ and $N2ATTACH$ indicate the attachment point (1-6) at nodes 1 and 2 at which the beam element frames into the two joints. $LENGTH$ indicates the clear length of the element between the two attachment points at the two nodes.

4. The strain assigned to each fiber of all segments of elastofiber and MEF elements, based on the user-specified probability distribution, are echoed in this section.

5. In addition to reflecting the input data provided by the user, this section provides the total number of panel zones, NPZ, computed by the program based on the joint restraint data provided by the user in the Nodal Data block. Note that each node could have a maximum of two panel zones, one in the $X'Z'$ plane of the ACJ and the other in the $X'Y'$ plane of the ACJ.

6. This section reflects the allowance provided by the user for the half-bandwidth of the stiffness matrix along with the half-bandwidth required for diaphragm elements and beam elements as computed internally by the program. If either of these two number exceeds the user-defined half-bandwidth, a warning is printed and the program is terminated. In such cases, increase the user-defined half-bandwidth to the greater of the program-computed half-bandwidths for plane stress elements and beam elements and rerun the analysis.

7. In this section, information pertaining to the static solution is provided. At each static load-step, the average and maximum number of elastofiber beam element local iterations performed for convergence to the member equilibrium are printed out during each global iteration. After global convergence is achieved for each load-step, the values of the $NRTH$ response quantities required by the user are printed out. Note that these time-histories are printed out sequentially in a separate output file, RTH, for convenient plotting (See Section 5.2).

8. The responses of all the beam elements and panel zone elements, along with nodal displacements, at the end of the static analysis are provided in this section.

   The beam element responses are provided in the following form:

   ELEM MPYL1 MYL1 MPYL2 MYL2 MPZL1 MZL1 MPZL2 MZL2 KAPYL1 KAPYL2 KAPZL1 KAPZL2 PY P PEPS MXL ALPHA ELEM

   Description

Variable	Definition
$ELEM$	Beam element number.
$MPYL1$	Plastic moment capacity at node 1 of the element about the local Y' (major) axis.
$MYL1$	Moment in the element at node 1 about the local Y' (major) axis.
$MPYL2$	Plastic moment capacity at node 2 of the element about the local Y' (major) axis.
$MYL2$	Moment in the element at node 2 about the local Y' (major) axis.
$MPZL1$	Plastic moment capacity at node 1 of the element about the local Z' (minor) axis.
$MZL1$	Moment in the element at node 1 about the local Z' (minor) axis.
$MPZL2$	Plastic moment capacity at node 2 of the element about the local Z' (minor) axis.
$MZL2$	Moment in the element at node 2 about the local Z' (minor) axis.
$KAPYL1$	Kink rotation in the element at node 1 about the local Y' (major) axis.
$KAPYL2$	Kink rotation in the element at node 2 about the local Y' (major) axis.
$KAPZL1$	Kink rotation in the element at node 1 about the local Z' (minor) axis.
$KAPZL2$	Kink rotation in the element at node 2 about the local Z' (minor) axis.
PY	Axial yield force of the element.
P	Axial force in the element.
$PEPS$	Axial strain in the element.
MXL	Torsion in the element.
$ALPHA$	Updated orientation angle of the element.
$ELEM$	Beam element number.

The panel zone element responses are provided in the following form (Note that data pertaining to the two panel zones at each joint are presented separately) :

NODE NPZZORY MY M PGAM

Description

Variable	Definition
$NODE$	Node number.
$NPZZORY$	Panel zone orientation: 1 for $X'Z'$ plane and 2 for $X'Y'$ plane.
MY	Panel zone yield moment.
M	Panel zone moment.
$PGAM$	Plastic shear strain in the panel zone.

The stresses in beam elements due to the static forces on the structure are provided in the following form:

ELEM YLMSTRESS ZLMSTRESS PSTRESS TSTRESS

Description

Variable	Definition
$ELEM$	Beam element number.
$YLMSTRESS$	Greater of the flexural stresses at the two ends of the element about the local Y' (major) axis.
$ZLMSTRESS$	Greater of the flexural stresses at the two ends of the element about the local Z' (minor) axis.
$PSTRESS$	Axial stress in the element.
$TSTRESS$	Total stress in the element.

The nodal displacements due to the static forces on the structure are provided in the following form:

NODE XDISP YDISP ZDISP

Description

Variable	Definition
$NODE$	Node number.
$XDISP$	Displacement of the node in the global X direction.
$YDISP$	Displacement of the node in the global Y direction.
$ZDISP$	Displacement of the node in the global Z direction.

9. In this section, the $NDSTPS$ data points of the three components of the ground acceleration specified by the user in input files $for090$, $for091$, and $for092$, are reproduced after being scaled by the user-defined scaling factors $GAMULTX$, $GAMULTY$, and $GAMULTZ$, respectively.

10. In this section, information pertaining to the dynamic solution is provided. At each dynamic time-step, the average and maximum number of elastofiber beam element local iterations performed for convergence to the member equilibrium are printed out during each global iteration. After global convergence is achieved for each time-step, the values of the $NRTH$ response quantities required by the user are printed out. Note that these time-histories are printed out sequentially in a separate output file, RTH, for convenient plotting (See Section 5.2).

11. The minimum (crest) and maximum (peak) responses of all the beam elements and panel zone elements at the end of the dynamic time-history analysis are provided in this section (this includes static response). The format and definition of all the output variables are the same as that provided in Note 7.

12. In this section, the plastic rotations (% radians) $YRPZ$ and $ZRPZ$, observed at the two panel zones of each joint during the dynamic time-history analysis, are provided. These plastic rotations are compared against FEMA-356 (FEMA 2000) acceptance criteria for Immediate Occupancy (IO), Life-Safety (LS), and Collapse Prevention (CP) performance levels and each panel zone is accordingly assigned a performance level.

13. In this section, the plastic rotations (% radians) $YROT1$, $ZROT1$, $YROT2$, and $ZROT2$, observed at the two nodes of the beam elements about the major (Y') and minor (Z') axes during the dynamic time-history analysis, are provided. These plastic rotations are compared against FEMA-356 (FEMA 2000) acceptance criteria for Immediate Occupancy (IO), Life-Safety (LS), and Collapse Prevention (CP) performance levels and each beam-end is accordingly assigned a performance level.

14. The structure performance is summarized in this section in the following two ways: (a) The number of structural components within each performance category is listed. (b) The number of structural components within various ranges of plastic rotations is listed. These ranges are $\leq$0.1%, (0.1-1.0]%, (1.0-2.0]%, (2.0-3.0]%, (3.0-4.0]%, (4.0-5.0]%, (5.0-6.0]%, (6.0-7.0]%, $>$ 7.0% of a radian.

15. In this section, the maximum and minimum principal stresses, and the maximum shear stresses that occur in the plane-stress elements during the course of the time-history analysis are listed along with the orientation of the maximum principal stress.

16. In this section, the damage index of each segment of elastofiber and MEF elements is presented. The damage index for a segment is defined as the average of the 20 fiber damage indices given by $\frac{\epsilon-\epsilon_y}{min(\epsilon_r,\epsilon_{frac})-\epsilon_y}$. The minimum value for the fiber damage index is taken to be 0.0 and the maximum value is taken to be 1.0. Average damage indices for all elastofiber and MEF elements are given as well, followed by the damage indices for all type 1 panel zones (with linear-quadratic shear stress-strain backbone relation), given by $\frac{\gamma-\gamma_y}{\gamma_u-\gamma_y}$, with limits of [0.0,1.0] imposed.

17. In this section, the peak interstory drift ratios during the course of the time-history analysis in the X and Y directions at the user-defined locations are summarized. The maximum drift ratio is compared

against FEMA-356 (FEMA 2000) acceptance criteria for Immediate Occupancy (IO), Life-Safety (LS), and Collapse Prevention (CP) performance levels, and the structure performance level is listed.

18. The time and location of the fibers that fracture in elastofiber beam element segments is summarized in this section.

5.2 File RTH

File RTH is an output file consisting of $NRTH$ columns with each column consisting of $NDSTPS$ data points of a user-defined response quantity, specified in the Time History Output Control Data block of input file, $for005$. This file, being fully numerical and consisting only of the data points with no additional characters, can be used for conveniently plotting the output time-histories of response quantities using a program such as Matlab.

5.3 File COORD

File $COORD$ is an output file consisting of 22 columns with each column consisting of NNP data points corresponding to the NNP node points in the structural model. Column 1 gives the node number. Columns 2, 3, and 4 give the nodal coordinates (X, Y, and Z). Columns 5 through 22 give the coordinates (X, Y and Z) of the 6 attachment points at each node. Thus columns 5, 6, and 7 give the coordinates of attachment point 1 and so on. Note that these coordinates correspond to the undeformed configuration of the structure. This file, being fully numerical and consisting only of the data points with no additional characters, can be used for conveniently plotting the undeformed shape of the structure.

5.4 File DEFCOORD

File $DEFCOORD$ is an output file consisting of the coordinates of the node points and the attachment points in the same format as file $COORD$, except that these coordinates are deformed shape coordinates written out at times $DTOUT$, $2 * DTOUT$, $3 * DTOUT$,...., $(NSSTPS + NDSTPS) * DT$, where $DTOUT$ is the time interval at which deformed shape coordinates are to be written out as specified by the user in the Deformed Shape Output Control Data block of input file $for005$. DT is the time-increment on the acceleration records. Thus, this file consists of $\frac{(NSSTPS+NDSTPS)*DT}{DTOUT}$ (rounded off to the nearest integer) sets of data with each set consisting of NNP lines. This file, being fully numerical and consisting only of the data points with no additional characters, can be used to conveniently plot the deformed shape of the structure at each time step. All these plots could then be strung together to create an animation of the structure shaking during an earthquake.

5.5 File BEAM

File $BEAM$ is an output file consisting of $NPHEL+NFIBEL+NFIBEL5$ lines of connectivity data for all the beam elements in the following form:

ELEM N1 N2 N1ATTACH N2ATTACH

where $N1$ and $N2$ correspond to the two nodes of the element while $N1ATTACH$ and $N2ATTACH$ correspond to the attachment points to which the element is physically connected at these nodes. This file, being fully numerical and consisting only of the data points with no additional characters, can be used for conveniently plotting the undeformed and deformed shapes of the structure.

5.6 File PLSTR

File $PLSTR$ is an output file consisting of $NUMPLSTREL$ lines of connectivity data for all the 4-noded diaphragm elements in the following form:

ELEM N1 N2 N3 N4

where $N1$, $N2$, $N3$, and $N4$, correspond to the four nodes of the element in that order. This file, being fully numerical and consisting only of the data points with no additional characters, can be used for conveniently plotting the undeformed and deformed shapes of the structure.

5.7 File BMPLROT

In this file, the plastic rotations (% radians) $YROT1$, $ZROT1$, $YROT2$, and $ZROT2$, observed at the two nodes of the beam elements about the major (Y') and minor (Z') axes during the dynamic time-history analysis, are provided in the following form:

ELEM X-N1ATTACH Y-N1ATTACH Z-N1ATTACH YROT1 ZROT1 X-N2ATTACH Y-N2ATTACH Z-N2ATTACH YROT2 ZROT2

where $X-N1ATTACH$, $Y-N1ATTACH$,and $Z-N1ATTACH$, are the final coordinates of the element attachment point at node 1 at the end of the analysis, while, $X-N2ATTACH$, $Y-N2ATTACH$,and $Z-N2ATTACH$, are the final coordinates of the element attachment point at node 2. This file, being fully numerical and consisting only of the data points with no additional characters, can be used for conveniently plotting plastic rotation maps of frames in the structure.

The file consists of $NPHEL+NFIBEL+NFIBEL5$ lines of data corresponding to all the beam elements in the structural model.

5.8 File PZPLROT

In this file, the plastic rotations (% radians) $YRPZ$ and $ZRPZ$, observed at the two panel zones of each joint during the dynamic time-history analysis, are provided in the following form:

NODE X Y Z YRPZ ZRPZ

where X, Y, and Z, are the coordinates of the node point at the end of the analysis. This file, being fully numerical and consisting only of the data points with no additional characters, can be used for conveniently plotting plastic rotation maps of frames in the structure.

The file consists of NNP lines of data corresponding to the NNP node points in the structural model.

5.9 File PLSTRSS

This file consists of principal stress-strain histories of the $NPLSTRSS$ diaphragm elements defined in the diaphragm element output control data block of input file, $for005$. For each time-step, output is presented in the following form:

PLSTRELEM σ_1 σ_2 θ_1 τ_{max}

where $PLSTRELEM$ corresponds to one of the diaphragm elements specified in the input file; σ_1 and σ_2 are the two principal stresses, θ_1 is the angle of the principal plane (corresponding to σ_1); and τ_{max} is the maximum shear stress in the element at that time-step.

5.10 File PLSTRSMAX

This file consists of the maximum and minimum principal stresses in each of the $NUMPLSTREL$ diaphragm elements during the course of the time-history analysis. The output is presented in the following form:

PLSTRELEM σ_1^{max} θ_1^{max} σ_2^{min} θ_2^{min} τ_{max}

where $PLSTRELEM$ corresponds to one of the $NUMPLSTREL$ plane stress elements in the model; σ_1^{max} is the maximum of σ_1 in the element for all time and θ_1^{max} is the corresponding orientation of the principal plane; σ_2^{min} is the minimum of σ_2 in the element for all time and θ_2^{min} is the corresponding orientation of the principal plane; and τ_{max} is the maximum shear stress in the element for all time.

5.11 File FRAC

This file lists the occurrence of fractures in the fibers of the elastofiber/MEF elements. The output is presented in the following form:

TIME ELEM SEGM FIBER

where $TIME$ is the product of the time-step and the time increment DT, at which the fiber fractures; $ELEM$ is the element ID; $SEGM$ is the segment ID (1 or 2 for elastofiber elements, 1 or 3 for end segments of MEF elements, and 2 for middle segments of MEF elements); and $FIBER$ is the fiber ID (1–20).

5.12 File FRACSUM

This file lists the elastofiber/MEF element segments with fracture in at least one fiber. It takes the results from file $FRAC$ and condenses all the fiber fractures within a single segment into a single line. The output is presented in the following form:

 # ELEM SEGM

where $ELEM$ refers to the elastofiber/MEF element ID and $SEGM$ is the segment ID (1 or 2).

5.13 File FRACTOT

This file gives the total number of segments of elastofiber/MEF elements with fracture in at least one fiber.

5.14 File RUP

This file lists the fiber ruptures in the elastofiber/MEF elements. The output is presented in the following form:

 # TIME ELEM SEGM FIBER

where $TIME$ is the product of the time-step and the time increment DT, at which the fiber ruptures; $ELEM$ is the element ID; $SEGM$ is the segment ID (1 or 2); and $FIBER$ is the fiber ID (1–20).

5.15 File FAIL

List of MEF/elastofiber beam element segments that completely fail. Complete failure of segment can occur when all the fibers either fracture or rupture. The output is presented in the following form:

TIME ELEM SEGM

where $TIME$ is the product of the time-step and the time increment DT, at which the fiber fractures; $ELEM$ is the element ID; and $SEGM$ is the segment ID (1 or 2).

5.16 File XDRFT

This file contains the peak interstory drifts in the X direction, corresponding to the user-defined input according to section 4.1.20. The output is presented in the following form:

 # XDRIFT

where XDRIFT is the X-direction peak interstory drift ratio during the time-history analysis.

5.17 File YDRFT

This file contains the peak interstory drifts in the Y direction, corresponding to the user-defined input according to section 4.1.20. The output is presented in the following form:

 # YDRIFT

where YDRIFT is the Y-direction peak interstory drift ratio during the time-history analysis.

5.18 File AVGDRFT

The average of the peak interstory drifts in the X and Y directions during the course of the time-history analysis are computed separately and the greater of the two numbers is listed in this file.

5.19 File XDRFTRTH

This file contains the X-direction interstory drift ratio time-histories of the $NDRIFTX$ user-requested IDRs. The file will consist of ($NSSTPS$ + $NDSTPS$) lines, each line consisting of $NDRIFTX$ drift quantities.

5.20 File YDRFTRTH

This file contains the Y-direction interstory drift ratio time-histories of the $NDRIFTY$ user-requested IDRs. The file will consist of ($NSSTPS$ + $NDSTPS$) lines, each line consisting of $NDRIFTY$ drift quantities.

5.21 File PKDRFT

The absolute peak interstory drift ratio in the model (X or Y direction) during the course of the time-history analysis is listed in this file.

5.22 File FEMA356

The plastic rotations at the ends of beams and columns, and in the panel zones are compared against FEMA356 (FEMA 2000) acceptance criteria for three performance levels, Immediate Occupancy (IO), Life-Safety (LS), and Collapse Prevention (CP). The number of connections at various performance levels are summarized in this file. If the plastic rotation at a connection exceeds the limits corresponding to these three performance levels, that connection is classified as having collapsed (CO). In addition, LS1 and LS2 are two intermediate performance levels that occur at the one-third and two-third points between the IO and LS performance levels.

Also summarized in this file are the number of panel zones and beam-column ends that have plastic rotations in various ranges, e.g., $\leq 0.1\%$, (0.1–1.0]%, (1.0–2.0]%, and so on.

5.23 File PERF

The overall performance of the building is assigned the performance level of the worst component (beam or column end or panel zone) and output in this file.

5.24 File EIGEN

The results from the subspace iteration method to evaluate a user-specified subset of Eigen values are presented in this file. The Eigen values are squares of the circular natural frequencies. The corresponding periods are also presented for convenience. Upper and lower bounds for the Eigen values are given as well, providing a guide to the user regarding the accuracy of the Eigen value estimation.

5.25 File MODES

This file contains the estimated $NEIG$ Eigen vectors determined using the subspace iteration methodology in column layout, with $NEIG$ columns, each column consisting of $NDOF$ ordinates corresponding to the N degrees of freedom in the system. To plot the mode shapes one would have to use these values in conjunction with the DOF numbers specified in output file $for006$.

5.26 File BUCKLING

This file contains the coordinates of the two interior nodes demarcating the middle segment of each 5-segment MEF element in the model for each time step. $NFIBEL5$ lines of data are provided in the following format for each time step:

ELEM X1 Y1 Z1 X2 Y2 Z2

where $ELEM$ is the element number; $X1$, $Y1$, and $Z1$ are the coordinates of the left end of the segment, and $X2$, $Y2$, and $Z2$ are the coordinates of the right end of the segment.

5.27 File ELMGRPRES

For each time step, the resultant forces in the global coordinate system for each of the $NGRP$ groups is output in this file. Thus, the file consists of $NGRP * (NSSTPS + NDSTPS)$ lines, $NGRP$ lines for each time step. The six columns correspond to three forces, F_X, F_Y, F_Z, and three moments, M_X, M_Y, and M_Z.

5.28 File PMM

The discretized PMM surface used for the plastic hinge elements in the model is output in this file. The file contains $NCURVE*3$ columns, each set of 3 columns consisting of the axial force, bending moment about

Y' axis and bending moment about Z' corresponding to each of the $NCURVES$. A total of $NPCURVE$ lines are provided for the $NPCURVE$ points on each curve.

5.29 File DAM_IND_EF

In this file, the damage index of each segment of all the elastofiber elements is presented. The damage index for a segment is defined as the average of the 20 fiber damage indices given by $\frac{\epsilon-\epsilon_y}{min(\epsilon_r,\epsilon_{frac})-\epsilon_y}$. The minimum value for the fiber damage index is taken to be 0.0 and the maximum value is taken to be 1.0. $NFIBEL$ lines of data are provided corresponding to the $NFIBEL$ lines in the following format:

IDFIBEL ELEM IDSEG DAM_IND

where $IDFIBEL$ is the elastofiber element number; $ELEM$ is the corresponding frame element number; $IDSEG$ is the segment number, 1 for the left segment, and 2 for the right segment; and DAM_IND is the damage index as defined above.

5.30 File DAM_IND_EF5_END

In this file, the damage index of of the two end segments of all the 5-segment MEF elements is presented. The damage index for a segment is defined as the average of the 20 fiber damage indices given by $\frac{\epsilon-\epsilon_y}{min(\epsilon_r,\epsilon_{frac})-\epsilon_y}$. The minimum value for the fiber damage index is taken to be 0.0 and the maximum value is taken to be 1.0. $NFIBEL5$ lines of data are provided corresponding to the $NFIBEL5$ lines in the following format:

IDFIBEL5 ELEM IDSEG DAM_IND

where $IDFIBEL5$ is the MEF element number; $ELEM$ is the corresponding frame element number; $IDSEG$ is the segment number, 1 for the left segment, and 3 for the right segment; and DAM_IND is the damage index as defined above.

5.31 File DAM_IND_EF5_MID

In this file, the damage index of the middle segment of all the 5-segment MEF elements is presented. The damage index for a segment is defined as the average of the 20 fiber damage indices given by $\frac{\epsilon-\epsilon_y}{min(\epsilon_r,\epsilon_{frac})-\epsilon_y}$. The minimum value for the fiber damage index is taken to be 0.0 and the maximum value is taken to be 1.0. $NFIBEL5$ lines of data are provided corresponding to the $NFIBEL5$ lines in the following format:

IDFIBEL5 ELEM IDSEG DAM_IND

where $IDFIBEL5$ is the MEF element number; $ELEM$ is the corresponding frame element number; $IDSEG$ is the segment number, 2 for the middle segment; and DAM_IND is the damage index as defined above.

5.32 File DAM_IND_PZ1

In this file, the damage indices for all type 1 panel zones (with linear-quadratic shear stress-strain backbone relation), given by $\frac{\gamma-\gamma_y}{\gamma_u-\gamma_y}$, with limits of [0.0,1.0] imposed, are provided in the following format:

IDPZ1 NODE NPZZORY DAM_IND

where $IDPZ1$ is the type 1 panel zone number; $NODE$ is the node number; $NPZZORY$ is the panel zone orientation, 1 for $X'Z'$ plane and 2 for $X'Y'$ plane; and DAM_IND is the damage index as defined above.

Chapter 6 Modeling and Trouble Shooting Tips

1. DATA CHECK: To check whether the input file is formatted correctly and that the program has read in the values of all input parameters, check the output file $for006$ where all the input data is echoed in the same order as occurs in the input file $for005$. If premature termination occurs, the last printed section in $for006$ should pin-point the location of error in the input files.

2. NO CONVERGENCE: Possible remedies for non-convergence of solution include the following.

   (a) Increase the maximum number of local and global iterations, $NLITMAX$ and $NGITMAX$, respectively.

   (b) Increase tolerance limits ($TOLF$, $TOLM$, $TOLFIBELF$, and $TOLFIBELM$).

   (c) If the model consists of elastofiber/MEF elements, ensure that the convergence tolerance limits for element local iteration, $TOLFIBELF$ and $TOLFIBELM$, are at least an order of magnitude smaller than the convergence tolerance limits for global model iteration, $TOLF$ and $TOLM$, respectively.

   (d) Try iterating with the original stiffness matrix by specifying a non-zero value for $TIMEOSTF$ (time until which iterations are performed using the original stiffness matrix; for subsequent time steps Newton-Raphson iterations will be conducted using the tangent stiffness matrix.

   (e) Try using a combination of the elastic stiffness matrix and the tangent stiffness matrix by specifying non-zero values for $STFFACPHE$ (plastic hinge elements), $STFFACFE$ (elastofiber/MEF elements), $STFFACPZ0$ (panel zone type I), and $STFFACPZ1$ (panel zone type II). **Warning: As elastofiber/MEF element fibers start rupturing, behavior becomes highly nonlinear. Member local iterations may not converge. If the local iterations for any member do not converge, a warning is issued in output file $for006$ and the analysis proceeds without termination. However, solution may be erroneous, especially after a few fibers have ruptured. For accurate solution, user will have to try various values of $STFFACFE$ until convergence is achieved in every member. Raising the convergence tolerance limit $TOLFIBELF$ might help.**

   (f) Reduce the size of loading in each step by subdividing the ground motion time history and correspondingly reducing the time step size, DT. Large loading steps may result in the structure alternating between two non-equilibrium states without ever converging to the true solution.

3. PROGRAM TERMINATES PREMATURELY (with a DIVIDE BY ZERO error in output file $FOR006$): The following are possible reasons for this.

   (a) Stiffness matrix has become ill-conditioned. This could be because the structure has become unstable and is collapsing or on the verge of collapsing. In many instances, the program will be able to follow through to complete collapse. In some instances it may terminate prematurely. Create an animation of the structural response to see whether this is indeed the case (using the deformed coordinates output in file $DEFCOORD$).

(b) A complete failure of a single element may in some instances result in stiffness-matrix ill-conditioning resulting in premature termination. In this case, iterating with the elastic stiffness or a multiple of the elastic stiffness may allow the analysis to proceed further. Try large non-zero values (0.1-3.0) for $STFFACPHE$ (plastic hinge elements), $STFFACFE$ (elastofiber/MEF elements), $STFFACPZ0$ (panel zone type I), and $STFFACPZ1$ (panel zone type II). **Warning: As elastofiber/MEF element fibers start rupturing, behavior becomes highly nonlinear. Member local iterations may not converge. If the local iterations for any member do not converge, a warning is issued in output file $for006$ and the analysis proceeds without termination. However, solution may be erroneous, especially after a few fibers have ruptured. For accurate solution, user will have to try various values of $STFFACFE$ until convergence is achieved in every member. Raising the convergence tolerance limit $TOLFIBELF$ might help.**

(c) For stiffening systems (e.g., braced frame systems where post-buckling straightening of braces during a tension excursion under cyclic loading loading may result in the system stiffening up), using a tangent stiffness may lead to divergence of the solution. Try using a value of 0.1 or larger for $STFFACFE$ (elastofiber/MEF elements). May to have to try very large values in some instances (1.0 or larger). **Warning: Note the word of caution in the last bullet point.**

(d) Try reducing the size of loading in each step by subdividing the ground motion time history and correspondingly reducing the time step size, DT.

4. For gravity columns that do not have any beams framing into them, specify a value of 0 for $IDASSCOL$. For gravity columns that have gravity beams framing into them, specify non-zero $IDASSCOL$. In this case clear length of column will be taken for stiffness computations.

5. Specify a minimum value of 0.000005 for $ECCMIN$ for all buckling-sensitive MEF elements.

6. Can model channel, angle, and T-shaped cross-sections by specifying a box or wide-flanged section, but selectively zeroing out the areas of certain fibers, effectively resulting in the desired cross-sectional shape. Use the $FAFRAC$ variable for this purpose.

Chapter 7   Examples

This chapter contains examples that were solved using Version 1.0 of the program. The examples have not been updated to reflect the changes to the input file as required by Versions 1.1 and 2.0 of the program.

7.1   Large Deflection of an Elastic Cantilever Beam

The problem of the large deflection of an elastic cantilever beam with a transversely acting point load at the free end has been analyzed by Mattiasson (1980). It has probably been the "test problem" most commonly used in the examination of finite element procedures for geometrically nonlinear beam and frame analysis. The cantilever beam is illustrated in Figure 7.1. Also illustrated on that figure is the comparison of the numerical solution by Mattiasson (1980) using elliptic integrals and the finite element solution using the elastofiber element. The beam is discretized into 10 elastofiber elements. The load, P, is increased gradually from 0 to 155.74 kN (35 Kips) and the horizontal (u) and vertical (w) deflections of the free end were computed. These displacements are normalized by the length, L, and plotted against the load, P, normalized by $\frac{EI}{L^2}$. The close match between the analytical and the FRAME3D solutions illustrates the ability of the elastofiber element to solve large deflection problems accurately.

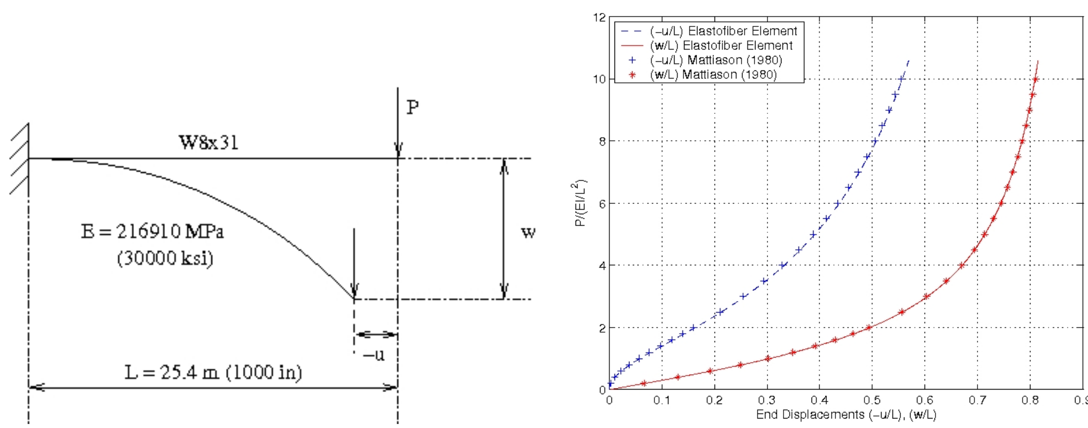


Figure 7.1: Large Deflection of a Cantilever Beam

The input file, $for005$, is shown in Figure 7.2.

```
CHEADING
Cantilever Beam
C NNP  NPHEL NFIBEL NPZ0 NPZ1 NSPR MTP NDIM MBD  NSSTPS  NDSTPS  NRTH  NGITMAX NLITMAX NOUT
  11    0    10    0    0    0  50  20000  6   1000    0    2    200    50    1
C  DT  TIMEOSTF BETA  GAMMA  A0    A1   AGRAV  STFFACPHE STFFACFE STFFACPZ0 STFFACPZ1 TOLF  TOLM   TOLFIBELF TOLFIBELM THRESH  NTHR  ITHR
  0.02 0.00    0.25   0.5 0.12566 0.0020372 386.4   0.01    0.00   0.01    0.01   0.01  0.1   0.01  0.1   200.   2    3
C  X    Y    Z  XRES YRES ZRES RXRES RYRES RZRES  FX   FY  FZ    MX    MY   MZ  IDASSCOL NODE #
  0.  0.   0.  0  0  0   0   0   0   0.   0.  0.   0.    0.  0.  0    1
 100.   0.   0.  1  0  1   0  1   0   0.   0.  0.000  0.000  0.   0.000 0     2
 200.   0.   0.  1  0  1   0  1   0   0.   0.  0.000  0.000  0.   0.000 0     3
 300.   0.   0.  1  0  1   0  1   0   0.   0.  0.000  0.000  0.   0.000 0     4
 400.   0.   0.  1  0  1   0  1   0   0.   0.  0.000  0.000  0.   0.000 0     5
 500.   0.   0.  1  0  1   0  1   0   0.   0.  0.000  0.000  0.   0.000 0     6
 600.   0.   0.  1  0  1   0  1   0   0.   0.  0.000  0.000  0.   0.000 0     7
 700.   0.   0.  1  0  1   0  1   0   0.   0.  0.000  0.000  0.   0.000 0     8
 800.   0.   0.  1  0  1   0  1   0   0.   0.  0.000  0.000  0.   0.000 0     9
 900.   0.   0.  1  0  1   0  1   0   0.   0.  0.000  0.000  0.   0.000 0     10
 1000.   0.   0.  1  0  1   0  1   0   0.   0. -35.000  0.000  0.   0.000 0     11
C NODE#  DOF#  SPRING CONSTANT (STIFFNESS)  ***********ELASTIC SPRING DATA***********
C NODE1 NODE2 IBC IDES   EE     HSH   SIGY  FAC1 FAC2 ALPHA IELTYPE OLFSRATIO  ES   SIGU   EPSS  EPSU  ICATFAFRAC ICATFYFRAC ELEM #
  1   2   1  8031  30000.  0.100   500000.  1.  1.   0.   1     0.20    30000.00 501000.000 0.80 1.00    1       1    1
  2   3   1  8031  30000.  0.100   500000.  1.  1.   0.   1     0.20    30000.00 501000.000 0.80 1.00    1       1    1
  3   4   1  8031  30000.  0.100   500000.  1.  1.   0.   1     0.20    30000.00 501000.000 0.80 1.00    1       1    1
  4   5   1  8031  30000.  0.100   500000.  1.  1.   0.   1     0.20    30000.00 501000.000 0.80 1.00    1       1    1
  5   6   1  8031  30000.  0.100   500000.  1.  1.   0.   1     0.20    30000.00 501000.000 0.80 1.00    1       1    1
  6   7   1  8031  30000.  0.100   500000.  1.  1.   0.   1     0.20    30000.00 501000.000 0.80 1.00    1       1    1
  7   8   1  8031  30000.  0.100   500000.  1.  1.   0.   1     0.20    30000.00 501000.000 0.80 1.00    1       1    1
  8   9   1  8031  30000.  0.100   500000.  1.  1.   0.   1     0.20    30000.00 501000.000 0.80 1.00    1       1    1
  9  10   1  8031  30000.  0.100   500000.  1.  1.   0.   1     0.20    30000.00 501000.000 0.80 1.00    1       1    1
 10  11   1  8031  30000.  0.100   500000.  1.  1.   0.   1     0.20    30000.00 501000.000 0.80 1.00    1       1    1
C ICAT IDSEG IDFIB(20)                              FAFRAC (END WITH LINE OF ZEROS)
  1   1   1  2  3  4  5  6  7  8  9 10 11 12 13 14 15 16 17 18 19 20 1.0
  1   2   1  2  3  4  5  6  7  8  9 10 11 12 13 14 15 16 17 18 19 20 1.0
  0   0   0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0 0.0
C ISEED (AN INTEGER BETWEEN 1 AND 1000000 TO GENERATE RANDOM NUMBERS FOR FRACTURE STRESS  COMPUTATION)
 999287
C ICAT IDSEG IDFIB(20)                              FYFRAC(10) (END WITH LINE OF ZEROS)
  1   1   1  2  3  4  5  6  7  8  9 10 11 12 13 14 15 16 17 18 19 20 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9
  1   2   1  2  3  4  5  6  7  8  9 10 11 12 13 14 15 16 17 18 19 20 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9
  0   0   0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0
C Note 1: If a -ve THK is input, web/flg thickness of ass. col. of corresponding node will be used for the PZ thickness
C Note 2: To select a particular beam as the beam governing the PZ depth, input the ID of the beam as a negative integer as HPZ
C NODE NPZZORY IPZTYPE G     HSH   TAUY   THK    DTHK    HPZ
C NUMPLSTREL NUMPLSTRMAT NINTPTPLSTR(=1 ==> 1-point Quadrature; =2 ==>2X2 Quadrature) NPLSTRSS
   0       0          20
C E     POIS PLSTRMATID      (FOR EACH OF THE NUMPLSTRMAT MATERIALS)
C FOR ELEMENTS N+NG, N+NG*2,..., NE-NG, NE, THE ELM MAT SET & THK NUMBERS ARE ALSO SET TO MAT(N) & THK(N) AND
C THE CONNECTIVITY ARRAYS ARE INCREMENTED BY LMG
C 0    0    0    0   0.00    0    0    0    0    0   TO END SERIES
C N   NE   NG   MAT#  THK   N1   N2   N3   N4   LMG
 0    0    0    0   0.00    0    0    0    0    0
C PRINCIPAL STRESS HISTORIES WILL BE WRITTEN OUT FOR PLANE STRESS ELEMENTS, N1 TO N2
C  N1   N2
C NODE RESPONSE: 1-U 2-V 3-W 4-THETAX 5-THETAY 7-THETAZ
C ELEM RESPONSE: 1-Mx 2-My1' 3-My2' 4-Mz1' 5-Mz2' 6-Ky1' 7-Ky2' 8-Kz1' 9-Kz2' 10-P 11-Epsilon
C NODE# RESPID# ELEM# RESPID# PZ# RESPID# ELEM# SEG# FIB# RESPID#
  11    1    0    0    0    0    0   0    0    0
  11    3    0    0    0    0    0   0    0    0
C    GAMULTX    GAMULTY    GAMULTZ
    0.3937    0.0000    0.3937
C TIME INTERVAL AT WHICH DEFORMED SHAPE COORDINATES HAVE TO BE PRINTED OUT (MAX # OF STATES = 250)
 0.10
```

Figure 7.2: Large Deflection of a Cantilever Beam: Input file, $for005$

The input file, $for009$, is shown in Figure 7.3.

IDES	AREA	SMI	WMI	SSM	WSM	SZ	WZ	DEPTH	WEBTHK	FLGWDTH	FLGTHK	TORJ
8031	9.13	110.	37.1	27.5	9.27	30.4	14.1	8.00	0.285	7.995	0.435	0.54

Figure 7.3: Large Deflection of a Cantilever Beam: Input file, $for009$

In this case, since dynamic time-history analysis is not required, input files $for090$, $for091$, and $for092$ are not required.

The output file, $for006$, is shown in Figures 7.4 through 7.7.

```
DATE 05/25/2003  TIME 19:01:30


Cantilever Beam

  NNP =  11
  NEL =  10
  NPHEL =   0
  NFIBEL =  10
  NPZ0 =   0
  NPZ1 =   0
  NSPR =   0
  MTP =   50
  NDIM =  20000
  MBD =   6
  NSSTPS =1000
  NDSTPS =   0
  NRTH =   2
  NGITMAX = 200
  NLITMAX =  50
  NOUT =   1

  DT = 0.20000E-01
  TIMEOSTF = 0.00000E+00
  BETA = 0.25000E+00
  GAMMA = 0.50000E+00
  A0 = 0.12566E+00
  A1 = 0.20372E-02
  AGRAV = 0.38640E+03
  STFFACPHE =  0.10000E-01
  STFFACFE =  0.00000E+00
  STFFACPZ0 =  0.10000E-01
  STFFACPZ1 =  0.10000E-01
  TOLF = 0.10000E-01
  TOLM = 0.10000E+00
  TOLFIBELF =  0.10000E-01
  TOLFIBELM =  0.10000E+00
  THRESH = 0.20000E+03
  NTHR =    2
  ITHR =  3
```

NODAL POINT INPUT

NODE	COORDX	COORDY	COORDZ	IDX	IDY	IDZ	IDTHX	IDTHY	IDTHZ	FX	FY	FZ	MX	MY	MZ	IDASSCOL
1	0.00000E+00	0.00000E+00	0.00000E+00	0	0	0	0	0	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
2	0.10000E+03	0.00000E+00	0.00000E+00	1	0	1	0	1	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
3	0.20000E+03	0.00000E+00	0.00000E+00	1	0	1	0	1	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
4	0.30000E+03	0.00000E+00	0.00000E+00	1	0	1	0	1	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
5	0.40000E+03	0.00000E+00	0.00000E+00	1	0	1	0	1	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
6	0.50000E+03	0.00000E+00	0.00000E+00	1	0	1	0	1	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
7	0.60000E+03	0.00000E+00	0.00000E+00	1	0	1	0	1	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
8	0.70000E+03	0.00000E+00	0.00000E+00	1	0	1	0	1	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
9	0.80000E+03	0.00000E+00	0.00000E+00	1	0	1	0	1	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
10	0.90000E+03	0.00000E+00	0.00000E+00	1	0	1	0	1	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
11	0.10000E+04	0.00000E+00	0.00000E+00	1	0	1	0	1	0	0.00000E+00	0.00000E+00	-0.35000E+02	0.00000E+00	0.00000E+00	0.00000E+00	0

```
*****************************

NDOF =   30

*****************************
```

DEGREE OF FREEDOM NUMBERS

NODE	DOFX	DOFY	DOFZ	DOFRX	DOFRYB	DOFRYC	DOFRZB	DOFRZC
1	0	0	0	0	0	0	0	0
2	1	0	2	0	3	3	0	0
3	4	0	5	0	6	6	0	0
4	7	0	8	0	9	9	0	0
5	10	0	11	0	12	12	0	0
6	13	0	14	0	15	15	0	0
7	16	0	17	0	18	18	0	0
8	19	0	20	0	21	21	0	0
9	22	0	23	0	24	24	0	0
10	25	0	26	0	27	27	0	0
11	28	0	29	0	30	30	0	0

NOTE THAT DOFX,Y,& Z REFER TO GLOBAL AXES
WHILE DOFRX,RYB,RYC,RZB,RZC ALL REFER TO PANEL ZONE
LOCAL AXES.

IF, HOWEVER, IDASSCOL OF A PARTICULAR NODE IS ZERO,
THEN THERE CANNOT BE ANY PANEL ZONES ASSOCIATED
WITH THAT NODE. HENCE, DOFRYC & DOFRZC ARE MEANINGLESS
AND THE ROTATIONS NOW REFER TO THE GLOBAL AXES
IN THIS CASE, i.e., THE PANEL ZONE COORDINATE SYSTEM
IS SYNONYMOUS WITH THE GLOBAL COORDINATE SYSTEM.

FOR GRAVITY COLUMNS THAT DO NOT HAVE ANY BEAMS
FRAMING INTO THEM, SPECIFY IDASSCOL=0.

FOR GRAVITY COLUMNS THAT HAVE GRAVITY BEAMS FRAMING
INTO THEM, SPECIFY NON-ZERO IDASSCOL. IN THIS CASE
CLEAR LENGTH OF COLUMN WILL BE TAKEN FOR STIFFNESS
COMPUTATIONS.

Figure 7.4: Large Deflection of a Cantilever Beam: Output file, $for006$

```
            ELASTIC SPRING DATA

SPRING# NODE# DOF#      SPRING STIFFNESS


                          JOINT DIMENSIONS

 NODE     H    D    W
   1 0.0000 0.0000 0.0000
   2 0.0000 0.0000 0.0000
   3 0.0000 0.0000 0.0000
   4 0.0000 0.0000 0.0000
   5 0.0000 0.0000 0.0000
   6 0.0000 0.0000 0.0000
   7 0.0000 0.0000 0.0000
   8 0.0000 0.0000 0.0000
   9 0.0000 0.0000 0.0000
  10 0.0000 0.0000 0.0000
  11 0.0000 0.0000 0.0000



                          ELEMENT INPUT

FIBER ELEMENT AREA REDUCTION CATEGORY INFORMATION
ICAT IDSEG  IDFIB                                     FAFRAC
  1  1  1  2  3  4  5  6  7  8  9 10 11 12 13 14 15 16 17 18 19 20 0.10000E+01
  1  2  1  2  3  4  5  6  7  8  9 10 11 12 13 14 15 16 17 18 19 20 0.10000E+01

ISEED =   999287

FIBER ELEMENT FRACTURE STRESS CATEGORY INFORMATION
ICAT IDSEG  IDFIB                                     FYFRAC(1) TO (10)
  1  1  1  2  3  4  5  6  7  8  9 10 11 12 13 14 15 16 17 18 19 20  0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10
  1  2  1  2  3  4  5  6  7  8  9 10 11 12 13 14 15 16 17 18 19 20  0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10


 ELEM  NODE1 N1ATTACH NODE2 N2ATTACH IBC  IDES     ALPHA      EE    HSH      SIGY    FAC1  FAC2   LENGTH IELTYPE OLFSRAT   ES        SIGU     EPSS     EPSU  ICATFAFRAC ICATFYFRAC
   1   1    0   2    0   1     8031   0. 0.30000E+05  0.1000 0.50000E+06 1.0000 1.0000100.0000  1   0.20   0.30000E+05 0.50100E+06  0.800  1.000   1    1
   2   2    0   3    0   1     8031   0. 0.30000E+05  0.1000 0.50000E+06 1.0000 1.0000100.0000  1   0.20   0.30000E+05 0.50100E+06  0.800  1.000   1    1
   3   3    0   4    0   1     8031   0. 0.30000E+05  0.1000 0.50000E+06 1.0000 1.0000100.0000  1   0.20   0.30000E+05 0.50100E+06  0.800  1.000   1    1
   4   4    0   5    0   1     8031   0. 0.30000E+05  0.1000 0.50000E+06 1.0000 1.0000100.0000  1   0.20   0.30000E+05 0.50100E+06  0.800  1.000   1    1
   5   5    0   6    0   1     8031   0. 0.30000E+05  0.1000 0.50000E+06 1.0000 1.0000100.0000  1   0.20   0.30000E+05 0.50100E+06  0.800  1.000   1    1
   6   6    0   7    0   1     8031   0. 0.30000E+05  0.1000 0.50000E+06 1.0000 1.0000100.0000  1   0.20   0.30000E+05 0.50100E+06  0.800  1.000   1    1
   7   7    0   8    0   1     8031   0. 0.30000E+05  0.1000 0.50000E+06 1.0000 1.0000100.0000  1   0.20   0.30000E+05 0.50100E+06  0.800  1.000   1    1
   8   8    0   9    0   1     8031   0. 0.30000E+05  0.1000 0.50000E+06 1.0000 1.0000100.0000  1   0.20   0.30000E+05 0.50100E+06  0.800  1.000   1    1
   9   9    0  10    0   1     8031   0. 0.30000E+05  0.1000 0.50000E+06 1.0000 1.0000100.0000  1   0.20   0.30000E+05 0.50100E+06  0.800  1.000   1    1
  10  10    0  11    0   1     8031   0. 0.30000E+05  0.1000 0.50000E+06 1.0000 1.0000100.0000  1   0.20   0.30000E+05 0.50100E+06  0.800  1.000   1    1

*****************************

NPZ =   0

*****************************



            PANEL ZONE INPUT

 PZ# NODE Z OR Y IPZTYPE   GG     HSH      TAUY   THK   DTHK   HPZ

PLANE STRESS ELEMENT DATA

NO.OF PLANE STRESS ELEMENTS, NUMPLSTREL =   0
NO.OF PLANE STRESS ELEMENT MATERIALS, NUMPLSTRMAT =   0
NO.OF INTEGRATION POINTS FOR GAUSS QUADRATURE,
NINTPTPLSTR =   2
NO.OF PLANE STRESS ELEMENTS, NUMPLSTRSS, FOR WHICH
PRINCIPAL STRESS HISTORY NEEDS TO BE WRITTEN OUT
IN FILE PLSTRSS =   0


  MATERIAL SETS
     SET          E        POIS

  PLANE STRESS ELEMENT THICKNESSES & MATERIAL SET       NUMBERS AND CONNECTIVITY MATRIX
     ELEM   MATSET   THK  NODE1  NODE2  NODE3  NODE4

PLANE STRESS ELEMENTS FOR WHICH PRINCIPAL STRESS
OUTPUT IS REQUIRED (FILE PLSTRSS)



*************************************************

MAX. BANDWIDTH INPUT BY USER           =   6
MAX. BANDWIDTH FOR PLANE STRESS ELEMENTS =   0
MAX. BANDWIDTH FOR FRAME ELEMENTS       =   6
USE THE GREATER OF THE ABOVE TWO BANDWIDTHS

*************************************************


 RESPONSE     IDRTH  (1)  (2)  (3)  (4)  (5)  (6)  (7)  (8)  (9)  (10)
    R 1          11    1    0    0    0    0    0    0    0    0
    R 2          11    3    0    0    0    0    0    0    0    0
```

Figure 7.5: Large Deflection of a Cantilever Beam: Output file, $for006$ (Contd.)

```
                    STATIC SOLUTION

AVE. NO.OF MEMBER ITER :   3
MAX. NO.OF MEMBER ITER :   4

AVE. NO.OF MEMBER ITER :   3
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   3
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   2

DATE 05/25/2003  TIME 19:01:31

 TIME = 0.020    NITER =  4
      R 1     R 2     R 3     R 4     R 5     R 6     R 7     R 8
  -0.15561E-01 -0.35636E+01

AVE. NO.OF MEMBER ITER :   3
MAX. NO.OF MEMBER ITER :   4

AVE. NO.OF MEMBER ITER :   3
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   3
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   2

DATE 05/25/2003  TIME 19:01:31

 TIME = 0.040    NITER =  4
      R 1     R 2     R 3     R 4     R 5     R 6     R 7     R 8
  -0.46362E-01 -0.71272E+01

AVE. NO.OF MEMBER ITER :   3
MAX. NO.OF MEMBER ITER :   4

AVE. NO.OF MEMBER ITER :   3
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   3
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   2

DATE 05/25/2003  TIME 19:01:32

 TIME = 0.060    NITER =  4
      R 1     R 2     R 3     R 4     R 5     R 6     R 7     R 8
  -0.92387E-01 -0.10690E+02
                    .
                    .
                    .
                    .
DATE 05/25/2003  TIME 19:07:22

 TIME =19.980    NITER =  11
      R 1     R 2     R 3     R 4     R 5     R 6     R 7     R 8
  -0.56959E+03 -0.81581E+03

AVE. NO.OF MEMBER ITER :   3
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   3
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   3
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   3

DATE 05/25/2003  TIME 19:07:22

 TIME =20.000    NITER =  17
      R 1     R 2     R 3     R 4     R 5     R 6     R 7     R 8
  -0.56981E+03 -0.81590E+03
```

Figure 7.6: Large Deflection of a Cantilever Beam: Output file, $for006$ (Contd.)

```
                    STATIC ELEMENT RESPONSES

DATE 05/25/2003  TIME 19:07:22

 ELEM   MPYL1       MYL1        MPYL2       MYL2        MPZL1       MZL1        MPZL2       MZL2        KAPYL1      KAPYL2      KAPZL1      KAPZL2      PY          P           PEPS        TORSION,MXL  ALPHA      ELEM
  1  0.15200E+08 -0.15074E+05 0.15200E+08 0.11672E+05 0.70500E+07 -0.13655E-11 0.70500E+07 -0.44334E-11 -0.90989E-01 0.74140E-01 -0.12213E-16 0.44191E-16 0.45650E+07 0.73849E+01 0.00000E+00 0.22874E-15 0.00000E+00  1
  2  0.15200E+08 -0.11672E+05 0.15200E+08 0.87368E+04 0.70500E+07 -0.23332E-11 0.70500E+07 0.34638E-11 -0.70171E-01 0.55655E-01 0.72510E-16 -0.77835E-16 0.45650E+07 0.18874E+02 0.00000E+00 0.65416E-16 0.00000E+00  2
  3  0.15200E+08 -0.87369E+04 0.15200E+08 0.64044E+04 0.70500E+07 0.28533E-11 0.70500E+07 -0.61535E-12 -0.52399E-01 0.40871E-01 -0.53554E-16 0.37470E-16 0.45650E+07 0.26016E+02 0.00000E+00 -0.65655E-16 0.00000E+00  3
  4  0.15200E+08 -0.64044E+04 0.15200E+08 0.46250E+04 0.70500E+07 -0.10462E-11 0.70500E+07 0.29898E-11 -0.38348E-01 0.29558E-01 0.47811E-16 -0.58579E-16 0.45650E+07 0.30103E+02 0.00000E+00 -0.23644E-16 0.00000E+00  4
  5  0.15200E+08 -0.46249E+04 0.15200E+08 0.32914E+04 0.70500E+07 0.15743E-11 0.70500E+07 0.11200E-11 -0.27657E-01 0.21070E-01 -0.14147E-16 -0.10519E-17 0.45650E+07 0.32343E+02 0.00000E+00 -0.44234E-16 0.00000E+00  5
  6  0.15200E+08 -0.32915E+04 0.15200E+08 0.22930E+04 0.70500E+07 -0.41678E-12 0.70500E+07 -0.33141E-12 -0.19649E-01 0.14719E-01 0.17984E-17 0.46137E-19 0.45650E+07 0.33538E+02 0.00000E+00 0.58381E-17 0.00000E+00  6
  7  0.15200E+08 -0.22931E+04 0.15200E+08 0.15349E+04 0.70500E+07 -0.22804E-12 0.70500E+07 -0.31311E-12 -0.13650E-01 0.99064E-02 0.17658E-17 0.13533E-17 0.45650E+07 0.34166E+02 0.00000E+00 0.39546E-17 0.00000E+00  7
  8  0.15200E+08 -0.15350E+04 0.15200E+08 0.94006E+03 0.70500E+07 0.99758E-12 0.70500E+07 0.30864E-12 -0.90837E-02 0.61464E-02 -0.10777E-16 0.44708E-17 0.45650E+07 0.34491E+02 0.00000E+00 -0.78462E-17 0.00000E+00  8
  9  0.15200E+08 -0.94012E+03 0.15200E+08 0.44601E+03 0.70500E+07 -0.69139E-13 0.70500E+07 -0.36406E-12 -0.54845E-02 0.30450E-02 -0.29294E-17 0.44199E-17 0.45650E+07 0.34651E+02 0.00000E+00 0.15333E-17 0.00000E+00  9
 10  0.15200E+08 -0.44608E+03 0.15200E+08 -0.36930E-01 0.70500E+07 0.69926E-13 0.70500E+07 -0.87044E-14 -0.24736E-02 0.27115E-03 -0.20258E-17 0.13892E-17 0.45650E+07 0.34717E+02 0.00000E+00 0.14589E-18 0.00000E+00 10

NODE NPZZORY   MY      M      PGAM

 ELEM YLMSTRESS  ZLMSTRESS  PSTRESS   TSTRESS
  1  0.54816E+03 0.47826E-12 0.80886E+00 0.54897E+03
  2  0.42443E+03 0.37366E-12 0.20672E+01 0.42650E+03
  3  0.31771E+03 0.30780E-12 0.28495E+01 0.32056E+03
  4  0.23289E+03 0.32252E-12 0.32972E+01 0.23618E+03
  5  0.16818E+03 0.16982E-12 0.35425E+01 0.17172E+03
  6  0.11969E+03 0.44960E-13 0.36734E+01 0.12336E+03
  7  0.83384E+02 0.33777E-13 0.37422E+01 0.87127E+02
  8  0.55817E+02 0.10761E-12 0.37778E+01 0.59594E+02
  9  0.34186E+02 0.39272E-13 0.37953E+01 0.37982E+02
 10  0.16221E+02 0.75432E-14 0.38026E+01 0.20024E+02

 NODE     XDISP       YDISP       ZDISP
  1  0.00000E+00 0.00000E+00 0.00000E+00
  2 -0.27880E+01 0.00000E+00 -0.21068E+02
  3 -0.18942E+02 0.00000E+00 -0.74943E+02
  4 -0.52336E+02 0.00000E+00 -0.14920E+03
  5 -0.10157E+03 0.00000E+00 -0.23509E+03
  6 -0.16354E+03 0.00000E+00 -0.32732E+03
  7 -0.23509E+03 0.00000E+00 -0.42291E+03
  8 -0.31350E+03 0.00000E+00 -0.52023E+03
  9 -0.39656E+03 0.00000E+00 -0.61845E+03
 10 -0.48250E+03 0.00000E+00 -0.71709E+03
 11 -0.56981E+03 0.00000E+00 -0.81590E+03
```

Figure 7.7: Large Deflection of a Cantilever Beam: Output file, $for006$ (Contd.)

7.2 2 Story Building with 2 Bays in Each Principal Direction

The second example consists of a 2-story building with 2 bays in either of the two principal directions. The model consists of 27 nodes, 30 beam elements, 8 diaphragm elements and 14 panel zone elements. Out of the 30 beam elements, 4 are gravity columns and are modeled using the plastic-hinge element while the remaining 26 are modeled using the elastofiber elements. The isometric view of the building model with node and beam element numbering is shown in Figure 7.8. The floor plan of the building with column orientations, location of moment framed connections and the diaphragm element numbering is illustrated in Figure 7.9. In addition to gravity loads, the building is subjected to ground shaking from the first 5 seconds of the Tabas strong-motion record from the 1978 Iran earthquake. The records are in units of g and hence a value of 386.4 $\frac{in}{sec^2}$ is used for the scaling factors, $GAMULTX$, $GAMULTY$, and $GAMULTZ$. Also, note that the variable $TIMEOSTF$ has been assigned a value of 4.00 sec. Thus, for the first 4 seconds of the record, the program uses the initial elastic stiffness matrix to iterate and after that it uses a tangent stiffness matrix for iteration.

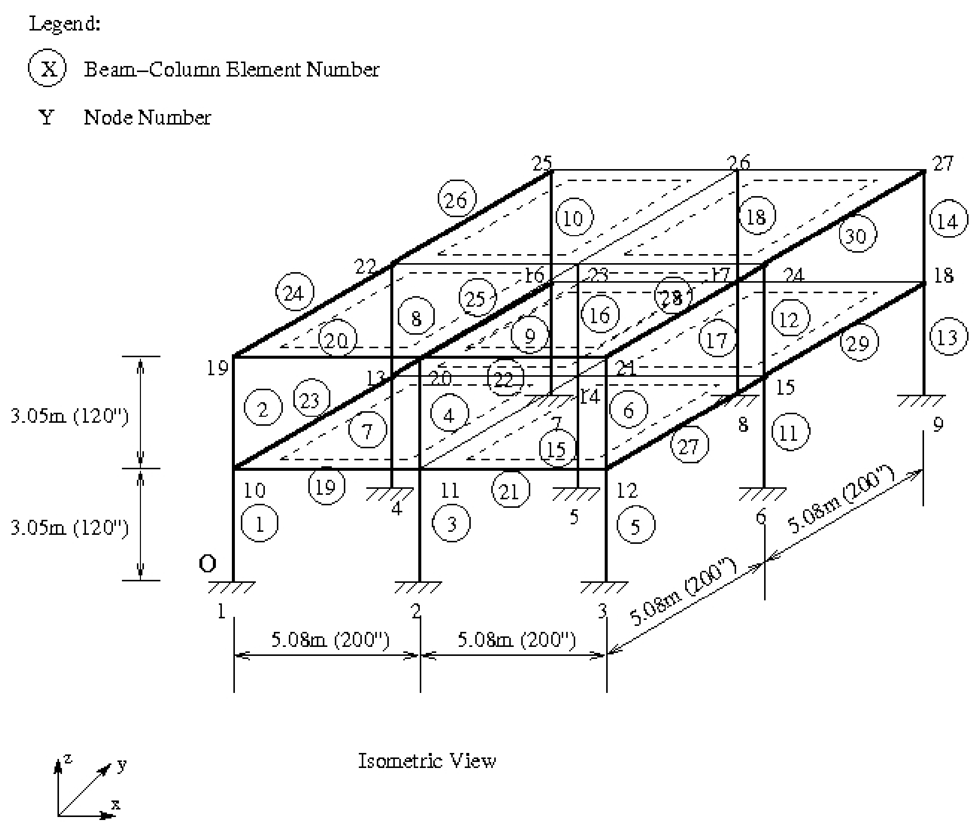


Figure 7.8: Isometric View of 2 Story Building

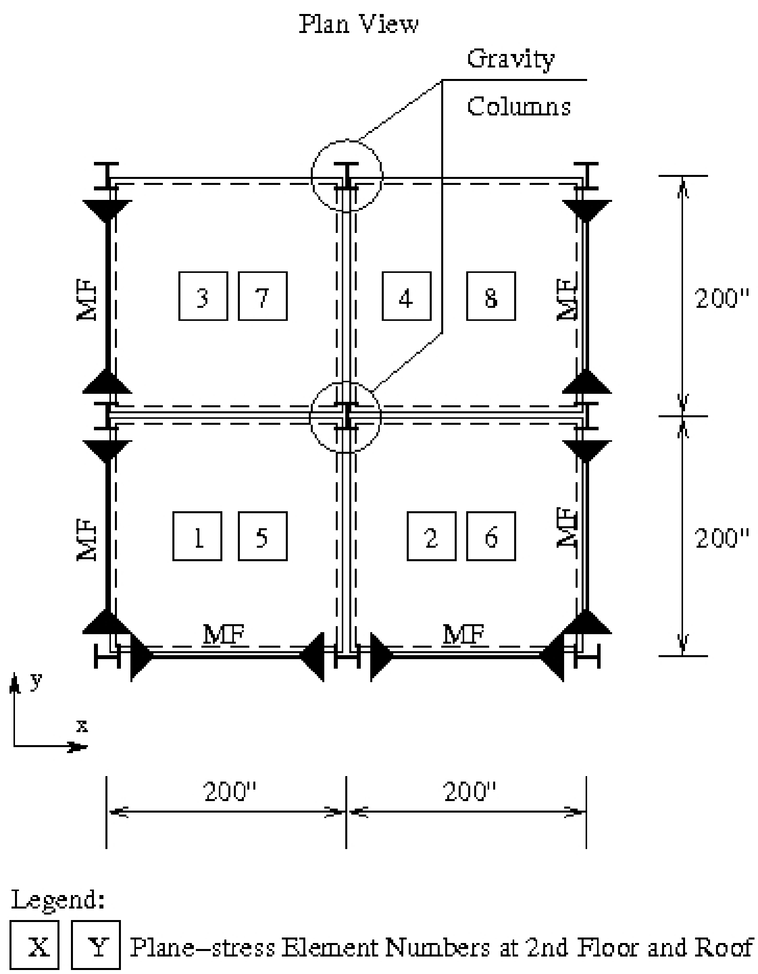


Figure 7.9: Plan View of 2 Story Building

The input file, $for005$, is shown in Figures 7.10 through 7.11.

```
CHEADING
2–Storeyed 3D 2–Bay x 2–Bay Building
C NNP  NPHEL NFIBEL NPZ0 NPZ1 NSPR MTP NDIM MBD  NSSTPS  NDSTPS  NRTH  NGITMAX NLITMAX NOUT
  27    4     26  0    18    0  40 10000 72  1      1000     3    100    50    1
C  DT  TIMEOSTF BETA  GAMMA  A0    A1   AGRAV  STFFACPHE STFFACFE STFFACPZ0 STFFACPZ1  TOLF  TOLM  TOLFIBELF TOLFIBELM THRESH  NTHR  ITHR
  0.005 4.00   0.25   0.5 0.12566 0.0020372 386.4   0.01    0.02   0.01    0.01 0.00001 0.00001 0.00001 0.00001  200.   2    3
C  X    Y    Z   XRES YRES ZRES RXRES RYRES RZRES  FX   FY   FZ    MX     MY    MZ  IDASSCOL NODE #
  0.   0.   0.  0  0  0   0   0   0   0.   0.  0.    0.    0.   0.   0     1
 200.   0.   0.  0  0  0   0   0   0   0.   0.  0.    0.    0.   0.   0     2
 400.   0.   0.  0  0  0   0   0   0   0.   0.  0.    0.    0.   0.   0     3
  0.  200.   0.  0  0  0   0   0   0   0.   0.  0.    0.    0.   0.   0     4
 200.  200.   0.  0  0  0   0   0   0   0.   0.  0.    0.    0.   0.   0     5
 400.  200.   0.  0  0  0   0   0   0   0.   0.  0.    0.    0.   0.   0     6
  0.  400.   0.  0  0  0   0   0   0   0.   0.  0.    0.    0.   0.   0     7
 200.  400.   0.  0  0  0   0   0   0   0.   0.  0.    0.    0.   0.   0     8
 400.  400.   0.  0  0  0   0   0   0   0.   0.  0.    0.    0.   0.   0     9
  0.   0.  120.  1  1  1   1   2   2   0.   0. –100.  100.   100. 100.   1    10
 200.   0.  120.  1  1  1   1   2   1   0.   0. –100.  100.   100. 100.   3    11
 400.   0.  120.  1  1  1   1   2   2   0.   0. –100.  100.   100. 100.   5    12
  0.  200.  120.  1  1  1   1   2   1   0.   0. –100.  100.   100. 100.   7    13
 200.  200.  120.  1  1  1   1   1   1   0.   0. –100.  100.   100. 100.   0    14
 400.  200.  120.  1  1  1   1   2   1   0.   0. –100.  100.   100. 100.  11    15
  0.  400.  120.  1  1  1   1   2   1   0.   0. –100.  100.   100. 100.   9    16
 200.  400.  120.  1  1  1   1   1   1   0.   0. –100.  100.   100. 100.   0    17
 400.  400.  120.  1  1  1   1   2   1   0.   0. –100.  100.   100. 100.  13    18
  0.   0.  240.  1  1  1   1   2   2   0.   0. –100.  100.   100. 100.   2    19
 200.   0.  240.  1  1  1   1   2   1   0.   0. –100.  100.   100. 100.   4    20
 400.   0.  240.  1  1  1   1   2   2   0.   0. –100.  100.   100. 100.   6    21
  0.  200.  240.  1  1  1   1   2   1   0.   0. –100.  100.   100. 100.   8    22
 200.  200.  240.  1  1  1   1   1   1   0.   0. –100.  100.   100. 100.   0    23
 400.  200.  240.  1  1  1   1   2   1   0.   0. –100.  100.   100. 100.  12    24
  0.  400.  240.  1  1  1   1   2   1   0.   0. –100.  100.   100. 100.  10    25
 200.  400.  240.  1  1  1   1   1   1   0.   0. –100.  100.   100. 100.   0    26
 400.  400.  240.  1  1  1   1   2   1   0.   0. –100.  100.   100. 100.  14    27
C NODE#  DOF#  SPRING CONSTANT (STIFFNESS)  ***********ELASTIC SPRING DATA***********
C NODE1 NODE2 IBC IDES   EE    HSH  SIGY  FAC1 FAC2 ALPHA IELTYPE OLFSRATIO  ES   SIGU   EPSS  EPSU  ICATFAFRAC ICATFYFRAC ELEM #
  1  10  2  14159 29000. 0.025  50.  1.  1.  0.   1   0.03   580.00 65.000 0.012 0.160   1     1   1
 10  19  2  14159 29000. 0.025  50.  1.  1.  0.   1   0.03   580.00 65.000 0.012 0.160   1     1   2
  2  11  2  14159 29000. 0.025  50.  1.  1.  0.   1   0.03   580.00 65.000 0.012 0.160   1     1   3
 11  20  2  14159 29000. 0.025  50.  1.  1.  0.   1   0.03   580.00 65.000 0.012 0.160   1     1   4
  3  12  2  14159 29000. 0.025  50.  1.  1.  0.   1   0.03   580.00 65.000 0.012 0.160   1     1   5
 12  21  2  14159 29000. 0.025  50.  1.  1.  0.   1   0.03   580.00 65.000 0.012 0.160   1     1   6
  4  13  2  14159 29000. 0.025  50.  1.  1.  90.  1   0.03   580.00 65.000 0.012 0.160   1     1   7
 13  22  2  14159 29000. 0.025  50.  1.  1.  90.  1   0.03   580.00 65.000 0.012 0.160   1     1   8
  7  16  2  14159 29000. 0.025  50.  1.  1.  90.  1   0.03   580.00 65.000 0.012 0.160   1     1   9
 16  25  2  14159 29000. 0.025  50.  1.  1.  90.  1   0.03   580.00 65.000 0.012 0.160   1     1  10
  6  15  2  14159 29000. 0.025  50.  1.  1.  90.  1   0.03   580.00 65.000 0.012 0.160   1     1  11
 15  24  2  14159 29000. 0.025  50.  1.  1.  90.  1   0.03   580.00 65.000 0.012 0.160   1     1  12
  9  18  2  14159 29000. 0.025  50.  1.  1.  90.  1   0.03   580.00 65.000 0.012 0.160   1     1  13
 18  27  2  14159 29000. 0.025  50.  1.  1.  90.  1   0.03   580.00 65.000 0.012 0.160   1     1  14
  5  14  2  14159 29000. 0.025  50.  1.  1.  90.  0   0.03   580.00 65.000 0.012 0.160   1     1  15
 14  23  2  14159 29000. 0.025  50.  1.  1.  90.  0   0.03   580.00 65.000 0.012 0.160   1     1  16
  8  17  2  14159 29000. 0.025  50.  1.  1.  90.  0   0.03   580.00 65.000 0.012 0.160   1     1  17
 17  26  2  14159 29000. 0.025  50.  1.  1.  90.  0   0.03   580.00 65.000 0.012 0.160   1     1  18
 10  11  1  24131 29000. 0.025  50.  1.  1.  0.   1   0.03   580.00 65.000 0.012 0.160   1     1  19
 19  20  1  24131 29000. 0.025  50.  1.  1.  0.   1   0.03   580.00 65.000 0.012 0.160   1     1  20
 11  12  1  24131 29000. 0.025  50.  1.  1.  0.   1   0.03   580.00 65.000 0.012 0.160   1     1  21
 20  21  1  24131 29000. 0.025  50.  1.  1.  0.   1   0.03   580.00 65.000 0.012 0.160   1     1  22
 10  13  1  24131 29000. 0.025  50.  1.  1.  0.   1   0.03   580.00 65.000 0.012 0.160   1     1  23
 19  22  1  24131 29000. 0.025  50.  1.  1.  0.   1   0.03   580.00 65.000 0.012 0.160   1     1  24
 13  16  1  24131 29000. 0.025  50.  1.  1.  0.   1   0.03   580.00 65.000 0.012 0.160   1     1  25
 22  25  1  24131 29000. 0.025  50.  1.  1.  0.   1   0.03   580.00 65.000 0.012 0.160   1     1  26
 12  15  1  24131 29000. 0.025  50.  1.  1.  0.   1   0.03   580.00 65.000 0.012 0.160   1     1  27
 21  24  1  24131 29000. 0.025  50.  1.  1.  0.   1   0.03   580.00 65.000 0.012 0.160   1     1  28
 15  18  1  24131 29000. 0.025  50.  1.  1.  0.   1   0.03   580.00 65.000 0.012 0.160   1     1  29
 24  27  1  24131 29000. 0.025  50.  1.  1.  0.   1   0.03   580.00 65.000 0.012 0.160   1     1  30
```

Figure 7.10: 2-Story Building: Input file, $for005$

```
C ICAT IDSEG IDFIB(20)                                  FAFRAC (END WITH LINE OF ZEROS)
  1   1   1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 1.0
  1   2   1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 1.0
  0   0   0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0.0
C ISEED (AN INTEGER BETWEEN 1 AND 1000000 TO GENERATE RANDOM NUMBERS FOR FRACTURE STRESS COMPUTATION)
 999287
C ICAT IDSEG IDFIB(20)                                  FYFRAC(10) (END WITH LINE OF ZEROS)
  1   1   1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9
  1   2   1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9
  0   0   0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0
C Note 1: If a -ve THK is input, web/flg thickness of ass. col. of corresponding node will be used for the PZ thickness
C Note 2: To select a particular beam as the beam governing the PZ depth, input the ID of the beam as a negative integer as HPZ
C NODE NPZZORY IPZTYPE G     HSH  TAUY  THK    DTHK    HPZ
  10    1     1 11600.  0.05  28.9  -1.0  0.000   -19
  10    2     1 11600.  0.05  28.9  -1.0  0.000   -23
  11    1     1 11600.  0.05  28.9  -1.0  0.000   -21
  12    1     1 11600.  0.05  28.9  -1.0  0.000   -21
  12    2     1 11600.  0.05  28.9  -1.0  0.000   -27
  13    1     1 11600.  0.05  28.9  -1.0  0.000   -23
  15    1     1 11600.  0.05  28.9  -1.0  0.000   -27
  16    1     1 11600.  0.05  28.9  -1.0  0.000   -25
  18    1     1 11600.  0.05  28.9  -1.0  0.000   -29
  19    1     1 11600.  0.05  28.9  -1.0  0.000   -20
  19    2     1 11600.  0.05  28.9  -1.0  0.000   -24
  20    1     1 11600.  0.05  28.9  -1.0  0.000   -20
  21    1     1 11600.  0.05  28.9  -1.0  0.000   -22
  21    2     1 11600.  0.05  28.9  -1.0  0.000   -28
  22    1     1 11600.  0.05  28.9  -1.0  0.000   -24
  24    1     1 11600.  0.05  28.9  -1.0  0.000   -28
  25    1     1 11600.  0.05  28.9  -1.0  0.000   -26
  27    1     1 11600.  0.05  28.9  -1.0  0.000   -30
C NUMPLSTREL NUMPLSTRMAT NINTPTPLSTR(=1 ==> 1-point Quadrature; =2 ==>2X2 Quadrature) NPLSTRSS
   8       1         20
C E     POIS PLSTRMATID      (FOR EACH OF THE NUMPLSTRMAT MATERIALS)
 3600.0  0.3   1
C FOR ELEMENTS N+NG, N+NG*2,..., NE-NG, NE, THE ELM MAT SET & THK NUMBERS ARE ALSO SET TO MAT(N) & THK(N) AND
C THE CONNECTIVITY ARRAYS ARE INCREMENTED BY LMG
C 0   0   0   0   0.00   0   0   0   0   0   TO END SERIES
C N   NE  NG   MAT#  THK   N1   N2   N3   N4   LMG
 1   2   1    1   3.5   10   11   14   13    1
 3   4   1    1   3.5   13   14   17   16    1
 5   6   1    1   3.5   19   20   23   22    1
 7   8   1    1   3.5   22   23   26   25    1
 0   0   0    0   0.0    0    0    0    0    0
C PRINCIPAL STRESS HISTORIES WILL BE WRITTEN OUT FOR PLANE STRESS ELEMENTS, N1 TO N2
C N1  N2
C NODE RESPONSE: 1-U 2-V 3-W 4-THETAX 5-THETAY 7-THETAZ
C ELEM RESPONSE: 1-Mx 2-My1' 3-My2' 4-Mz1' 5-Mz2' 6-Ky1' 7-Ky2' 8-Kz1' 9-Kz2' 10-P 11-Epsilon
C NODE# RESPID# ELEM# RESPID# PZ# RESPID# ELEM# SEG# FIB# RESPID#
   27    1     0     0    0    0     0   0   0   0
   27    2     0     0    0    0     0   0   0   0
   27    3     0     0    0    0     0   0   0   0
C    GAMULTX    GAMULTY    GAMULTZ
   386.400   386.400   386.400
C TIME INTERVAL AT WHICH DEFORMED SHAPE COORDINATES HAVE TO BE PRINTED OUT (MAX # OF STATES = 250)
 0.10
```

Figure 7.11: 2-Story Building: Input file, $for005$ (Contd.)

The input file, $for009$, is shown in Figure 7.12.

IDES	AREA	SMI	WMI	SSM	WSM	SZ	WZ	DEPTH	WEBTHK	FLGWDTH	FLGTHK	TORJ
14159	46.43	1900.	748.	254.	96.2	287.	146.0	14.98	0.745	15.565	1.190	19.80
24131	38.5	4020.	340.	329.	53.0	370.	81.5	24.48	0.605	12.855	0.960	9.50
24146	43.0	4580.	391.	371.	60.5	418.	93.2	24.74	0.650	12.900	1.090	13.40
24162	47.7	5170.	443.	414.	68.4	468.	105.0	25.00	0.705	12.955	1.220	18.50

Figure 7.12: 2-Story Building: Input file, $for009$

The output file, $for006$, is shown in Figures 7.13 through 7.23.

DATE 06/20/2003 TIME 19:44:04

2–Storeyed 3D 2–Bay x 2–Bay Building

NNP = 27
NEL = 30
NPHEL = 4
NFIBEL = 26
NPZ0 = 0
NPZ1 = 18
NSPR = 0
MTP = 40
NDIM = 10000
MBD = 72
NSSTPS = 1
NDSTPS =1000
NRTH = 3
NGITMAX = 100
NLITMAX = 50
NOUT = 1

DT = 0.50000E–02
TIMEOSTF = 0.40000E+01
BETA = 0.25000E+00
GAMMA = 0.50000E+00
A0 = 0.12566E+00
A1 = 0.20372E–02
AGRAV = 0.38640E+03
STFFACPHE = 0.10000E–01
STFFACFE = 0.20000E–01
STFFACPZ0 = 0.10000E–01
STFFACPZ1 = 0.10000E–01
TOLF = 0.10000E–04
TOLM = 0.10000E–04
TOLFIBELF = 0.10000E–04
TOLFIBELM = 0.10000E–04
THRESH = 0.20000E+03
NTHR = 2
ITHR = 3

NODAL POINT INPUT

NODE	COORDX	COORDY	COORDZ	IDX	IDY	IDZ	IDTHX	IDTHY	IDTHZ	FX	FY	FZ	MX	MY	MZ	IDASSCOL
1	0.00000E+00	0.00000E+00	0.00000E+00	0	0	0	0	0	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
2	0.20000E+03	0.00000E+00	0.00000E+00	0	0	0	0	0	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
3	0.40000E+03	0.00000E+00	0.00000E+00	0	0	0	0	0	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
4	0.00000E+00	0.20000E+03	0.00000E+00	0	0	0	0	0	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
5	0.20000E+03	0.20000E+03	0.00000E+00	0	0	0	0	0	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
6	0.40000E+03	0.20000E+03	0.00000E+00	0	0	0	0	0	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
7	0.00000E+00	0.40000E+03	0.00000E+00	0	0	0	0	0	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
8	0.20000E+03	0.40000E+03	0.00000E+00	0	0	0	0	0	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
9	0.40000E+03	0.40000E+03	0.00000E+00	0	0	0	0	0	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
10	0.00000E+00	0.00000E+00	0.12000E+03	1	1	1	1	2	2	0.00000E+00	0.00000E+00	–0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	1
11	0.20000E+03	0.00000E+00	0.12000E+03	1	1	1	1	2	1	0.00000E+00	0.00000E+00	–0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	3
12	0.40000E+03	0.00000E+00	0.12000E+03	1	1	1	1	2	2	0.00000E+00	0.00000E+00	–0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	5
13	0.00000E+00	0.20000E+03	0.12000E+03	1	1	1	1	2	1	0.00000E+00	0.00000E+00	–0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	7
14	0.20000E+03	0.20000E+03	0.12000E+03	1	1	1	1	1	1	0.00000E+00	0.00000E+00	–0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	0
15	0.40000E+03	0.20000E+03	0.12000E+03	1	1	1	1	2	1	0.00000E+00	0.00000E+00	–0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	11
16	0.00000E+00	0.40000E+03	0.12000E+03	1	1	1	1	2	1	0.00000E+00	0.00000E+00	–0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	9
17	0.20000E+03	0.40000E+03	0.12000E+03	1	1	1	1	1	1	0.00000E+00	0.00000E+00	–0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	0
18	0.40000E+03	0.40000E+03	0.12000E+03	1	1	1	1	2	1	0.00000E+00	0.00000E+00	–0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	13
19	0.00000E+00	0.00000E+00	0.24000E+03	1	1	1	1	2	2	0.00000E+00	0.00000E+00	–0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	2
20	0.20000E+03	0.00000E+00	0.24000E+03	1	1	1	1	2	1	0.00000E+00	0.00000E+00	–0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	4
21	0.40000E+03	0.00000E+00	0.24000E+03	1	1	1	1	2	2	0.00000E+00	0.00000E+00	–0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	6
22	0.00000E+00	0.20000E+03	0.24000E+03	1	1	1	1	2	1	0.00000E+00	0.00000E+00	–0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	8
23	0.20000E+03	0.20000E+03	0.24000E+03	1	1	1	1	1	1	0.00000E+00	0.00000E+00	–0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	0
24	0.40000E+03	0.20000E+03	0.24000E+03	1	1	1	1	2	1	0.00000E+00	0.00000E+00	–0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	12
25	0.00000E+00	0.40000E+03	0.24000E+03	1	1	1	1	2	1	0.00000E+00	0.00000E+00	–0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	10
26	0.20000E+03	0.40000E+03	0.24000E+03	1	1	1	1	1	1	0.00000E+00	0.00000E+00	–0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	0
27	0.40000E+03	0.40000E+03	0.24000E+03	1	1	1	1	2	1	0.00000E+00	0.00000E+00	–0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	14

Figure 7.13: 2-Story Building: Output file, $for006$

```
******************************

NDOF =   126

******************************



DEGREE OF FREEDOM NUMBERS

NODE  DOFX  DOFY  DOFZ  DOFRX DOFRYB DOFRYC DOFRZB DOFRZC
  1    0    0    0    0    0    0    0    0
  2    0    0    0    0    0    0    0    0
  3    0    0    0    0    0    0    0    0
  4    0    0    0    0    0    0    0    0
  5    0    0    0    0    0    0    0    0
  6    0    0    0    0    0    0    0    0
  7    0    0    0    0    0    0    0    0
  8    0    0    0    0    0    0    0    0
  9    0    0    0    0    0    0    0    0
 10    1    2    3    4    5    6    7    8
 11    9   10   11   12   13   14   15   15
 12   16   17   18   19   20   21   22   23
 13   24   25   26   27   28   29   30   30
 14   31   32   33   34   35   35   36   36
 15   37   38   39   40   41   42   43   43
 16   44   45   46   47   48   49   50   50
 17   51   52   53   54   55   55   56   56
 18   57   58   59   60   61   62   63   63
 19   64   65   66   67   68   69   70   71
 20   72   73   74   75   76   77   78   78
 21   79   80   81   82   83   84   85   86
 22   87   88   89   90   91   92   93   93
 23   94   95   96   97   98   98   99   99
 24  100  101  102  103  104  105  106  106
 25  107  108  109  110  111  112  113  113
 26  114  115  116  117  118  118  119  119
 27  120  121  122  123  124  125  126  126


NOTE THAT DOFX,Y,& Z REFER TO GLOBAL AXES
WHILE DOFRX,RYB,RYC,RZB,RZC ALL REFER TO PANEL ZONE
LOCAL AXES.

IF, HOWEVER, IDASSCOL OF A PARTICULAR NODE IS ZERO,
THEN THERE CANNOT BE ANY PANEL ZONES ASSOCIATED
WITH THAT NODE. HENCE, DOFRYC & DOFRZC ARE MEANINGLESS
AND THE ROTATIONS NOW REFER TO THE GLOBAL AXES
IN THIS CASE, i.e., THE PANEL ZONE COORDINATE SYSTEM
IS SYNONYMOUS WITH THE GLOBAL COORDINATE SYSTEM.

FOR GRAVITY COLUMNS THAT DO NOT HAVE ANY BEAMS
FRAMING INTO THEM, SPECIFY IDASSCOL=0.

FOR GRAVITY COLUMNS THAT HAVE GRAVITY BEAMS FRAMING
INTO THEM, SPECIFY NON-ZERO IDASSCOL. IN THIS CASE
CLEAR LENGTH OF COLUMN WILL BE TAKEN FOR STIFFNESS
COMPUTATIONS.



         ELASTIC SPRING DATA

SPRING# NODE# DOF#      SPRING STIFFNESS
```

Figure 7.14: 2-Story Building: Output file, $for006$ (Contd.)

```
                    JOINT DIMENSIONS

NODE     H     D     W
  1  0.0000  0.0000  0.0000
  2  0.0000  0.0000  0.0000
  3  0.0000  0.0000  0.0000
  4  0.0000  0.0000  0.0000
  5  0.0000  0.0000  0.0000
  6  0.0000  0.0000  0.0000
  7  0.0000  0.0000  0.0000
  8  0.0000  0.0000  0.0000
  9  0.0000  0.0000  0.0000
 10 24.4800 14.9800 15.5650
 11 24.4800 14.9800 15.5650
 12 24.4800 14.9800 15.5650
 13 24.4800 14.9800 15.5650
 14  0.0000  0.0000  0.0000
 15 24.4800 14.9800 15.5650
 16 24.4800 14.9800 15.5650
 17  0.0000  0.0000  0.0000
 18 24.4800 14.9800 15.5650
 19 24.4800 14.9800 15.5650
 20 24.4800 14.9800 15.5650
 21 24.4800 14.9800 15.5650
 22 24.4800 14.9800 15.5650
 23  0.0000  0.0000  0.0000
 24 24.4800 14.9800 15.5650
 25 24.4800 14.9800 15.5650
 26  0.0000  0.0000  0.0000
 27 24.4800 14.9800 15.5650



                    ELEMENT INPUT

FIBER ELEMENT AREA REDUCTION CATEGORY INFORMATION
ICAT IDSEG  IDFIB                                             FAFRAC
  1  1  1  2  3  4  5  6  7  8  9 10 11 12 13 14 15 16 17 18 19 20 0.10000E+01
  1  2  1  2  3  4  5  6  7  8  9 10 11 12 13 14 15 16 17 18 19 20 0.10000E+01

ISEED =   999287

FIBER ELEMENT FRACTURE STRESS CATEGORY INFORMATION
ICAT IDSEG  IDFIB                                             FYFRAC(1) TO (10)
  1  1  1  2  3  4  5  6  7  8  9 10 11 12 13 14 15 16 17 18 19 20  0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10
  1  2  1  2  3  4  5  6  7  8  9 10 11 12 13 14 15 16 17 18 19 20  0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10


 ELEM  NODE1 N1ATTACH NODE2 N2ATTACH IBC  IDES    ALPHA     EE    HSH      SIGY   FAC1  FAC2   LENGTH IELTYPE OLFSRAT  ES        SIGU    EPSS    EPSU ICATFAFRAC ICATFYFRAC
  1   1    0  10    6   2    14159   0. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000107.7600  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
  2  10    5  19    6   2    14159   0. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000 95.5200  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
  3   2    0  11    6   2    14159   0. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000107.7600  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
  4  11    5  20    6   2    14159   0. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000 95.5200  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
  5   3    0  12    6   2    14159   0. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000107.7600  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
  6  12    5  21    6   2    14159   0. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000 95.5200  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
  7   4    0  13    6   2    14159  90. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000107.7600  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
  8  13    5  22    6   2    14159  90. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000 95.5200  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
  9   7    0  16    6   2    14159  90. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000107.7600  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 10  16    5  25    6   2    14159  90. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000 95.5200  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 11   6    0  15    6   2    14159  90. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000107.7600  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 12  15    5  24    6   2    14159  90. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000 95.5200  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 13   9    0  18    6   2    14159  90. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000107.7600  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 14  18    5  27    6   2    14159  90. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000 95.5200  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 15   5    0  14    0   2    14159  90. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000120.0000  0   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 16  14    0  23    0   2    14159  90. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000120.0000  0   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 17   8    0  17    0   2    14159  90. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000120.0000  0   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 18  17    0  26    0   2    14159  90. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000120.0000  0   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 19  10    1  11    2   1    24131   0. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000185.0200  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 20  19    1  20    2   1    24131   0. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000185.0200  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 21  11    1  12    2   1    24131   0. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000185.0200  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 22  20    1  21    2   1    24131   0. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000185.0200  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 23  10    4  13    2   1    24131   0. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000184.7275  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 24  19    4  22    2   1    24131   0. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000184.7275  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 25  13    1  16    2   1    24131   0. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000185.0200  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 26  22    1  25    2   1    24131   0. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000185.0200  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 27  12    4  15    2   1    24131   0. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000184.7275  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 28  21    4  24    2   1    24131   0. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000184.7275  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 29  15    1  18    2   1    24131   0. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000185.0200  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
 30  24    1  27    2   1    24131   0. 0.29000E+05  0.0250 0.50000E+02  1.0000  1.0000185.0200  1   0.03   0.58000E+03  0.65000E+02  0.012  0.160   1    1
```

Figure 7.15: 2-Story Building: Output file, $for006$ (Contd.)

****

NPZ = 18

****

PANEL ZONE INPUT

PZ#	NODE	Z OR Y	IPZTYPE	GG	HSH	TAUY	THK	DTHK	HPZ
1	10	1	1	0.11600E+05	0.0500	0.28900E+02	0.7450	0.0000	24.4800
2	10	2	1	0.11600E+05	0.0500	0.28900E+02	2.3800	0.0000	24.4800
3	11	1	1	0.11600E+05	0.0500	0.28900E+02	0.7450	0.0000	24.4800
4	12	1	1	0.11600E+05	0.0500	0.28900E+02	0.7450	0.0000	24.4800
5	12	2	1	0.11600E+05	0.0500	0.28900E+02	2.3800	0.0000	24.4800
6	13	1	1	0.11600E+05	0.0500	0.28900E+02	0.7450	0.0000	24.4800
7	15	1	1	0.11600E+05	0.0500	0.28900E+02	0.7450	0.0000	24.4800
8	16	1	1	0.11600E+05	0.0500	0.28900E+02	0.7450	0.0000	24.4800
9	18	1	1	0.11600E+05	0.0500	0.28900E+02	0.7450	0.0000	24.4800
10	19	1	1	0.11600E+05	0.0500	0.28900E+02	0.7450	0.0000	24.4800
11	19	2	1	0.11600E+05	0.0500	0.28900E+02	2.3800	0.0000	24.4800
12	20	1	1	0.11600E+05	0.0500	0.28900E+02	0.7450	0.0000	24.4800
13	21	1	1	0.11600E+05	0.0500	0.28900E+02	0.7450	0.0000	24.4800
14	21	2	1	0.11600E+05	0.0500	0.28900E+02	2.3800	0.0000	24.4800
15	22	1	1	0.11600E+05	0.0500	0.28900E+02	0.7450	0.0000	24.4800
16	24	1	1	0.11600E+05	0.0500	0.28900E+02	0.7450	0.0000	24.4800
17	25	1	1	0.11600E+05	0.0500	0.28900E+02	0.7450	0.0000	24.4800
18	27	1	1	0.11600E+05	0.0500	0.28900E+02	0.7450	0.0000	24.4800

PLANE STRESS ELEMENT DATA

NO.OF PLANE STRESS ELEMENTS, NUMPLSTREL = 8
NO.OF PLANE STRESS ELEMENT MATERIALS, NUMPLSTRMAT = 1
NO.OF INTEGRATION POINTS FOR GAUSS QUADRATURE,
NINTPTPLSTR = 2
NO.OF PLANE STRESS ELEMENTS, NUMPLSTRSS, FOR WHICH
PRINCIPAL STRESS HISTORY NEEDS TO BE WRITTEN OUT
IN FILE PLSTRSS = 0

MATERIAL SETS

SET	E	POIS
1	0.36000E+04	0.30000E+00

PLANE STRESS ELEMENT THICKNESSES & MATERIAL SET NUMBERS AND CONNECTIVITY MATRIX

ELEM	MATSET	THK	NODE1	NODE2	NODE3	NODE4
1	1	3.500	10	11	14	13
2	1	3.500	11	12	15	14
3	1	3.500	13	14	17	16
4	1	3.500	14	15	18	17
5	1	3.500	19	20	23	22
6	1	3.500	20	21	24	23
7	1	3.500	22	23	26	25
8	1	3.500	23	24	27	26

PLANE STRESS ELEMENTS FOR WHICH PRINCIPAL STRESS
OUTPUT IS REQUIRED (FILE PLSTRSS)

****

MAX. BANDWIDTH INPUT BY USER = 72
MAX. BANDWIDTH FOR PLANE STRESS ELEMENTS = 36
MAX. BANDWIDTH FOR FRAME ELEMENTS = 71
USE THE GREATER OF THE ABOVE TWO BANDWIDTHS

****

RESPONSE	IDRTH	(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
R 1	27	1	0	0	0	0	0	0	0	0	
R 2	27	2	0	0	0	0	0	0	0	0	
R 3	27	3	0	0	0	0	0	0	0	0	

Figure 7.16: 2-Story Building: Output file, $for006$ (Contd.)

STATIC SOLUTION

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 2

DATE 06/20/2003 TIME 19:44:05

TIME = 0.005 NITER = 1

R 1	R 2	R 3	R 4	R 5	R 6	R 7	R 8
0.30731E-16	-0.19949E-16	-0.23100E-01					

STATIC ELEMENT RESPONSES

DATE 06/20/2003 TIME 19:44:05

ELEM	MPYL1	MYL1	MPYL2	MYL2	MPZL1	MZL1	MPZL2	MZL2	KAPYL1	KAPYL2	KAPZL1	KAPZL2	PY	P	PEPS	TORSION,MXL	ALPHA	ELEM
1	0.14350E+05	-0.27200E-13	0.14350E+05	-0.83512E-13	0.73000E+04	-0.38395E-15	0.73000E+04	-0.67345E-15	0.51352E-21	0.42082E-20	-0.29931E-20	-0.83226E-21	0.23215E+04	-0.20000E+03	0.00000E+00	0.10972E-16	-0.14748E-18	1
2	0.14350E+05	0.11289E-12	0.14350E+05	0.10991E-13	0.73000E+04	-0.36491E-13	0.73000E+04	-0.58525E-13	0.78972E-20	0.24612E-20	-0.57439E-20	-0.43226E-20	0.23215E+04	-0.10000E+03	0.00000E+00	0.34036E-16	-0.70046E-18	2
3	0.14350E+05	-0.85650E-13	0.14350E+05	-0.28806E-13	0.73000E+04	0.14267E-13	0.73000E+04	-0.14501E-13	-0.12765E-20	-0.19639E-21	-0.19739E-20	0.24662E-20	0.23215E+04	-0.20000E+03	0.00000E+00	0.11284E-16	-0.15167E-18	3
4	0.14350E+05	0.14226E-12	0.14350E+05	0.15353E-15	0.73000E+04	0.14270E-13	0.73000E+04	0.62312E-14	0.30718E-20	-0.25505E-20	-0.19906E-20	0.26005E-21	0.23215E+04	-0.10000E+03	0.00000E+00	0.31725E-16	-0.68131E-18	4
5	0.14350E+05	0.27880E-13	0.14350E+05	-0.54200E-15	0.73000E+04	0.13548E-13	0.73000E+04	-0.15333E-13	-0.19809E-20	-0.95238E-22	-0.49150E-20	-0.16807E-20	0.23215E+04	-0.20000E+03	0.00000E+00	0.13298E-16	-0.17874E-18	5
6	0.14350E+05	-0.18044E-13	0.14350E+05	-0.27739E-12	0.73000E+04	-0.79680E-13	0.73000E+04	-0.59507E-13	-0.11179E-20	-0.11898E-19	-0.92865E-20	-0.66269E-20	0.23215E+04	-0.10000E+03	0.00000E+00	0.27540E-16	-0.68559E-18	6
7	0.14350E+05	0.30531E-13	0.14350E+05	0.94480E-13	0.73000E+04	0.14154E-13	0.73000E+04	-0.14900E-13	-0.40953E-20	0.12175E-19	-0.40993E-20	0.34307E-20	0.23215E+04	-0.20000E+03	0.00000E+00	0.10499E-16	0.90000E+02	7
8	0.14350E+05	0.18312E-12	0.14350E+05	0.38207E-12	0.73000E+04	-0.69268E-14	0.73000E+04	0.57000E-14	0.12205E-19	0.18489E-19	-0.26053E-20	0.16764E-21	0.23215E+04	-0.10000E+03	0.00000E+00	0.33323E-16	0.90000E+02	8
9	0.14350E+05	-0.31360E-12	0.14350E+05	-0.29378E-13	0.73000E+04	0.14196E-13	0.73000E+04	-0.14928E-13	-0.76959E-20	0.40328E-20	-0.44021E-20	0.41365E-20	0.23215E+04	-0.20000E+03	0.00000E+00	0.10473E-16	0.90000E+02	9
10	0.14350E+05	-0.14264E-12	0.14350E+05	-0.36646E-12	0.73000E+04	-0.35355E-13	0.73000E+04	-0.87272E-14	-0.78419E-20	-0.18100E-19	-0.31972E-20	0.23232E-21	0.23215E+04	-0.10000E+03	0.00000E+00	0.35885E-16	0.90000E+02	10
11	0.14350E+05	0.31242E-13	0.14350E+05	0.13072E-12	0.73000E+04	0.14158E-13	0.73000E+04	0.14158E-13	-0.40875E-20	0.14891E-19	-0.40848E-20	0.35616E-20	0.23215E+04	-0.20000E+03	0.00000E+00	0.13233E-16	0.90000E+02	11
12	0.14350E+05	0.18336E-12	0.14350E+05	0.39652E-12	0.73000E+04	-0.74005E-14	0.73000E+04	0.68104E-14	0.12523E-19	0.18976E-19	-0.27176E-20	0.14402E-21	0.23215E+04	-0.10000E+03	0.00000E+00	0.25123E-16	0.90000E+02	12
13	0.14350E+05	-0.44135E-12	0.14350E+05	-0.29234E-13	0.73000E+04	0.14116E-13	0.73000E+04	-0.14306E-13	-0.83564E-20	0.52441E-20	-0.47582E-20	0.38202E-20	0.23215E+04	-0.20000E+03	0.00000E+00	0.11925E-16	0.90000E+02	13
14	0.14350E+05	-0.13304E-12	0.14350E+05	-0.17923E-12	0.73000E+04	-0.35774E-13	0.73000E+04	-0.24722E-15	-0.60796E-20	-0.11392E-19	-0.29644E-20	0.80813E-21	0.23215E+04	-0.10000E+03	0.00000E+00	0.41397E-16	0.90000E+02	14
15	0.14350E+05	-0.39121E-13	0.14350E+05	0.27953E-13	0.73000E+04	-0.30004E-13	0.73000E+04	0.15059E-13	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.23215E+04	-0.20000E+03	0.00000E+00	0.00000E+00	0.90000E+02	15
16	0.14350E+05	-0.28060E-13	0.14350E+05	0.98560E-15	0.73000E+04	-0.15245E-13	0.73000E+04	0.13953E-15	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.23215E+04	-0.10000E+03	0.00000E+00	0.00000E+00	0.90000E+02	16
17	0.14350E+05	-0.38312E-13	0.14350E+05	0.27951E-13	0.73000E+04	-0.33188E-13	0.73000E+04	0.18134E-13	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.23215E+04	-0.20000E+03	0.00000E+00	0.00000E+00	0.90000E+02	17
18	0.14350E+05	-0.28052E-13	0.14350E+05	0.55364E-14	0.73000E+04	-0.18345E-13	0.73000E+04	-0.24556E-15	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.23215E+04	-0.10000E+03	0.00000E+00	0.00000E+00	0.90000E+02	18
19	0.18500E+05	-0.51337E-12	0.18500E+05	0.12832E-11	0.40750E+04	0.45795E-16	0.40750E+04	-0.12223E-15	-0.57769E-20	0.40879E-19	-0.35222E-23	-0.12473E-22	0.19250E+04	-0.69823E-15	0.00000E+00	-0.21466E-16	0.31766E-17	19
20	0.18500E+05	0.37127E-13	0.18500E+05	-0.94078E-12	0.40750E+04	0.15751E-15	0.40750E+04	-0.10522E-15	0.20836E-19	-0.65912E-20	0.15574E-22	0.65499E-22	0.19250E+04	0.94719E-15	0.00000E+00	-0.28951E-16	0.50467E-17	20
21	0.18500E+05	-0.28017E-12	0.18500E+05	0.12454E-11	0.40750E+04	0.10554E-15	0.40750E+04	-0.17686E-15	-0.15348E-19	0.36927E-19	0.36104E-23	-0.54249E-22	0.19250E+04	-0.11084E-14	0.00000E+00	0.87718E-17	0.37872E-17	21
22	0.18500E+05	0.20669E-12	0.18500E+05	-0.37883E-12	0.40750E+04	0.27246E-15	0.40750E+04	-0.79534E-16	0.90472E-20	-0.57594E-20	0.26987E-22	0.75731E-22	0.19250E+04	0.11164E-14	0.00000E+00	0.27739E-17	0.63057E-17	22
23	0.18500E+05	0.79928E-13	0.18500E+05	0.34715E-13	0.40750E+04	0.60232E-16	0.40750E+04	-0.17135E-15	-0.15057E-19	0.41043E-20	-0.14215E-22	-0.62551E-24	0.19250E+04	0.48350E-15	0.00000E+00	-0.38032E-16	0.53184E-17	23
24	0.18500E+05	0.25530E-12	0.18500E+05	-0.12813E-12	0.40750E+04	0.16617E-15	0.40750E+04	-0.28891E-15	-0.55938E-20	-0.20102E-19	-0.20677E-23	0.29691E-22	0.19250E+04	-0.81912E-15	0.00000E+00	-0.53736E-16	0.73688E-17	24
25	0.18500E+05	-0.25857E-12	0.18500E+05	0.56951E-12	0.40750E+04	0.14428E-15	0.40750E+04	-0.22023E-15	-0.10232E-19	0.83011E-20	-0.15397E-22	-0.14640E-22	0.19250E+04	-0.25247E-14	0.00000E+00	-0.99256E-17	0.76222E-17	25
26	0.18500E+05	-0.76602E-12	0.18500E+05	-0.15233E-13	0.40750E+04	0.28110E-15	0.40750E+04	-0.57559E-15	-0.10837E-19	-0.38136E-20	-0.34955E-22	-0.99482E-22	0.19250E+04	0.23710E-14	0.00000E+00	-0.16605E-16	0.10748E-16	26
27	0.18500E+05	0.49573E-12	0.18500E+05	-0.47691E-12	0.40750E+04	0.24441E-15	0.40750E+04	-0.88332E-16	0.60446E-20	-0.18561E-19	0.44591E-22	0.46452E-22	0.19250E+04	0.84333E-15	0.00000E+00	-0.33387E-17	0.70004E-17	27
28	0.18500E+05	-0.21874E-12	0.18500E+05	-0.26885E-12	0.40750E+04	0.42739E-15	0.40750E+04	-0.27543E-15	-0.27172E-19	-0.25529E-19	-0.24725E-22	0.38743E-22	0.19250E+04	-0.14540E-14	0.00000E+00	0.39612E-16	0.11743E-16	28
29	0.18500E+05	-0.83845E-12	0.18500E+05	0.10150E-11	0.40750E+04	0.72466E-16	0.40750E+04	-0.22679E-15	-0.16023E-19	0.31575E-19	-0.56188E-22	-0.18607E-22	0.19250E+04	-0.18096E-14	0.00000E+00	-0.90735E-17	0.75972E-17	29
30	0.18500E+05	-0.32722E-12	0.18500E+05	0.48837E-13	0.40750E+04	0.31601E-15	0.40750E+04	-0.10813E-14	-0.81679E-20	0.41691E-20	-0.87473E-23	-0.38588E-21	0.19250E+04	0.17989E-14	0.00000E+00	-0.17941E-16	0.10704E-16	30

NODE	NPZZORY	MY	M	PGAM
10	1	0.63164E+04	-0.20808E-12	0.00000E+00
10	2	0.20966E+05	-0.28582E-13	0.00000E+00
11	1	0.63164E+04	-0.27118E-12	0.12037E-34
12	1	0.63164E+04	-0.22922E-13	0.00000E+00
12	2	0.20966E+05	-0.98635E-13	0.00000E+00
13	1	0.63164E+04	0.12668E-12	0.60185E-35
15	1	0.63164E+04	0.40725E-13	0.15046E-35
16	1	0.63164E+04	0.58048E-13	0.00000E+00
18	1	0.63164E+04	0.11164E-13	0.00000E+00
19	1	0.63164E+04	-0.28985E-12	0.00000E+00
19	2	0.20966E+05	-0.59698E-13	0.00000E+00
20	1	0.63164E+04	-0.44631E-12	0.00000E+00
21	1	0.63164E+04	-0.46878E-13	0.00000E+00
21	2	0.20966E+05	-0.15971E-12	0.00000E+00
22	1	0.63164E+04	0.24300E-12	0.12037E-34
24	1	0.63164E+04	0.25168E-12	0.00000E+00
25	1	0.63164E+04	0.64624E-13	-0.30093E-35
27	1	0.63164E+04	-0.52931E-13	0.00000E+00

Figure 7.17: 2-Story Building: Output file, $for006$ (Contd.)

ELEM	YLMSTRESS	ZLMSTRESS	PSTRESS	TSTRESS
1	0.32879E−15	0.70005E−17	0.43076E+01	0.43076E+01
2	0.44446E−15	0.60837E−15	0.21538E+01	0.21538E+01
3	0.33720E−15	0.15074E−15	0.43076E+01	0.43076E+01
4	0.56009E−15	0.14834E−15	0.21538E+01	0.21538E+01
5	0.10976E−15	0.15938E−15	0.43076E+01	0.43076E+01
6	0.10921E−14	0.82827E−15	0.21538E+01	0.21538E+01
7	0.37197E−15	0.15489E−15	0.43076E+01	0.43076E+01
8	0.15042E−14	0.72005E−16	0.21538E+01	0.21538E+01
9	0.12346E−14	0.15518E−15	0.43076E+01	0.43076E+01
10	0.14428E−14	0.36751E−15	0.21538E+01	0.21538E+01
11	0.51464E−15	0.14717E−15	0.43076E+01	0.43076E+01
12	0.15611E−14	0.76928E−16	0.21538E+01	0.21538E+01
13	0.17376E−14	0.14871E−15	0.43076E+01	0.43076E+01
14	0.70562E−15	0.37187E−15	0.21538E+01	0.21538E+01
15	0.15402E−15	0.31189E−15	0.43076E+01	0.43076E+01
16	0.11047E−15	0.15847E−15	0.21538E+01	0.21538E+01
17	0.15083E−15	0.34499E−15	0.43076E+01	0.43076E+01
18	0.11044E−15	0.19070E−15	0.21538E+01	0.21538E+01
19	0.39002E−14	0.23063E−17	0.18136E−16	0.39207E−14
20	0.28595E−14	0.29718E−17	0.24602E−16	0.28871E−14
21	0.37856E−14	0.33370E−17	0.28790E−16	0.38177E−14
22	0.11515E−14	0.51407E−17	0.28996E−16	0.11856E−14
23	0.24294E−15	0.32329E−17	0.12558E−16	0.25873E−15
24	0.77600E−15	0.54511E−17	0.21276E−16	0.80273E−15
25	0.17310E−14	0.41553E−17	0.65577E−16	0.18008E−14
26	0.23283E−14	0.10860E−16	0.61584E−16	0.24008E−14
27	0.15068E−14	0.46115E−17	0.21905E−16	0.15333E−14
28	0.81718E−15	0.80639E−17	0.37767E−16	0.86301E−15
29	0.30852E−14	0.42790E−17	0.47002E−16	0.31365E−14
30	0.99459E−15	0.20402E−16	0.46723E−16	0.10617E−14

NODE	XDISP	YDISP	ZDISP
1	0.00000E+00	0.00000E+00	0.00000E+00
2	0.00000E+00	0.00000E+00	0.00000E+00
3	0.00000E+00	0.00000E+00	0.00000E+00
4	0.00000E+00	0.00000E+00	0.00000E+00
5	0.00000E+00	0.00000E+00	0.00000E+00
6	0.00000E+00	0.00000E+00	0.00000E+00
7	0.00000E+00	0.00000E+00	0.00000E+00
8	0.00000E+00	0.00000E+00	0.00000E+00
9	0.00000E+00	0.00000E+00	0.00000E+00
10	0.74394E−17	−0.36494E−17	−0.16006E−01
11	0.73238E−17	−0.46873E−17	−0.16006E−01
12	0.71402E−17	−0.58130E−17	−0.16006E−01
13	0.84428E−17	−0.35694E−17	−0.16006E−01
14	0.84328E−17	−0.47312E−17	−0.17824E−01
15	0.84333E−17	−0.56734E−17	−0.16006E−01
16	0.94119E−17	−0.39878E−17	−0.16006E−01
17	0.94896E−17	−0.46542E−17	−0.17824E−01
18	0.95750E−17	−0.59733E−17	−0.16006E−01
19	0.23211E−16	−0.12509E−16	−0.23100E−01
20	0.23368E−16	−0.16335E−16	−0.23100E−01
21	0.23553E−16	−0.20007E−16	−0.23100E−01
22	0.27042E−16	−0.12645E−16	−0.23100E−01
23	0.27004E−16	−0.16235E−16	−0.26737E−01
24	0.26990E−16	−0.20247E−16	−0.23100E−01
25	0.30836E−16	−0.12251E−16	−0.23100E−01
26	0.30817E−16	−0.16215E−16	−0.26737E−01
27	0.30731E−16	−0.19949E−16	−0.23100E−01

Figure 7.18: 2-Story Building: Output file, $for006$ (Contd.)

```
                        TIME HISTORY INPUT
NDSTPS =   1000 DT =     5.0000000000000D-03

 TIME HISTORY SCALING FACTORS

 GAMULTX  GAMULTY  GAMULTZ
  386.4000 386.4000 386.4000

  SCALED X GROUND ACCELERATION SPACED AT DT
    0.0000    2.6426    5.2852    7.9278   10.5700    9.5858    8.6020    7.6179    6.6337    5.9324
    5.2311    4.5298    3.8286    3.8110    3.7934    3.7758    3.7582    3.9069    4.0560    4.2048
    4.3536    3.0861    1.8187    0.5513   -0.7161    0.5117    1.7394    2.9671    4.1948    5.2477
    6.3003    7.3532    8.4057    6.3683    4.3304    2.2928    0.2551   -0.4361   -1.1273   -1.8186
   -2.5098   -1.1714    0.1671    1.5055    2.8439    3.1352    3.4264    3.7177    4.0089    3.4434
                             •
                             •
                             •
                             •

  SCALED Y GROUND ACCELERATION SPACED AT DT
    0.0000    2.7024    5.4046    8.1071   10.8095    9.3049    7.8006    6.2960    4.7917    3.6156
    2.4396    1.2635    0.0875   -0.4822   -1.0518   -1.6215   -2.1911   -2.3798   -2.5684   -2.7571
   -2.9457   -4.7118   -6.4776   -8.2435  -10.0093   -8.4181   -6.8269   -5.2353   -3.6441   -3.2422
   -2.8402   -2.4382   -2.0363   -2.6935   -3.3507   -4.0077   -4.6650   -5.7098   -6.7547   -7.7995
   -8.8443   -7.5657   -6.2875   -5.0089   -3.7305   -3.1314   -2.5324   -1.9334   -1.3344   -2.3694
                             •
                             •
                             •
                             •

  SCALED Z GROUND ACCELERATION SPACED AT DT
    0.0000   -2.4571   -4.9142   -7.3714   -9.8285   -8.9761   -8.1233   -7.2705   -6.4181   -4.8798
   -3.3417   -1.8035   -0.2653   -1.5429   -2.8204   -4.0982   -5.3756   -5.7817   -6.1882   -6.5943
   -7.0004   -5.9312   -4.8617   -3.7925   -2.7232   -3.6239   -4.5247   -5.4254   -6.3261   -4.7489
   -3.1715   -1.5941   -0.0168   -1.4324   -2.8480   -4.2635   -5.6793   -4.9911   -4.3030   -3.6149
   -2.9268   -3.0216   -3.1164   -3.2111   -3.3058   -2.2805   -1.2552   -0.2299    0.7954    2.0045
                        DYNAMIC SOLUTION


DATE 06/20/2003  TIME 19:44:05

 TIME = 0.010    NITER =   0    ZT = 0.00000E+00
        R 1      R 2      R 3      R 4      R 5      R 6      R 7      R 8
   0.30731E-16 -0.19949E-16 -0.23100E-01

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   2

DATE 06/20/2003  TIME 19:44:05

 TIME = 0.015    NITER =   2    ZT = 0.00000E+00
        R 1      R 2      R 3      R 4      R 5      R 6      R 7      R 8
  -0.16540E-04 -0.16939E-04 -0.23087E-01

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   2

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   2

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   2

DATE 06/20/2003  TIME 19:44:05

 TIME = 0.020    NITER =   4    ZT = 0.00000E+00
        R 1      R 2      R 3      R 4      R 5      R 6      R 7      R 8
  -0.99321E-04 -0.10178E-03 -0.23024E-01
```

Figure 7.19: 2-Story Building: Output file, $for006$ (Contd.)

```
AVE. NO.OF MEMBER ITER :   3
MAX. NO.OF MEMBER ITER :   4

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   2

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   2

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   2

DATE 06/20/2003  TIME 19:44:05

 TIME = 0.025    NITER =   4    ZT = 0.00000E+00
       R 1      R 2      R 3      R 4      R 5      R 6      R 7      R 8
   -0.31488E-03 -0.32290E-03 -0.22887E-01
                    •
                    •
                    •
                    •
DATE 06/20/2003  TIME 19:51:54

 TIME = 5.000    NITER =  20    ZT = 0.00000E+00
       R 1      R 2      R 3      R 4      R 5      R 6      R 7      R 8
    0.36033E+01  0.55247E-02 -0.55692E-01

AVE. NO.OF MEMBER ITER :   6
MAX. NO.OF MEMBER ITER :  12

AVE. NO.OF MEMBER ITER :   5
MAX. NO.OF MEMBER ITER :  10

AVE. NO.OF MEMBER ITER :   4
MAX. NO.OF MEMBER ITER :   9

AVE. NO.OF MEMBER ITER :   4
MAX. NO.OF MEMBER ITER :   8

AVE. NO.OF MEMBER ITER :   4
MAX. NO.OF MEMBER ITER :   8

AVE. NO.OF MEMBER ITER :   3
MAX. NO.OF MEMBER ITER :   8

AVE. NO.OF MEMBER ITER :   3
MAX. NO.OF MEMBER ITER :   7

AVE. NO.OF MEMBER ITER :   3
MAX. NO.OF MEMBER ITER :   7

AVE. NO.OF MEMBER ITER :   3
MAX. NO.OF MEMBER ITER :   6

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   6

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   5

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   5

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   4

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   4

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   3

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   2

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   2

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   2

AVE. NO.OF MEMBER ITER :   2
MAX. NO.OF MEMBER ITER :   2

DATE 06/20/2003  TIME 19:51:56

 TIME = 5.005    NITER =  20    ZT = 0.00000E+00
       R 1      R 2      R 3      R 4      R 5      R 6      R 7      R 8
    0.38069E+01 -0.88064E-01 -0.57891E-01
```

Figure 7.20: 2-Story Building: Output file, $for006$ (Contd.)

MINIMUM STATIC PLUS DYNAMIC ELEMENT RESPONSES

DATE 06/20/2003 TIME 19:51:56

ELEM	MPYL1	MYL1	MPYL2	MYL2	MPZL1	MZL1	MPZL2	MZL2	KAPYL1	KAPYL2	KAPZL1	KAPZL2	PY	P	PEPS	TORSION,MXL	ALPHA	ELEM
1	0.14350E+05	−0.91422E+04	0.14350E+05	−0.31924E+04	0.73000E+04	−0.31255E+04	0.73000E+04	−0.21229E+04	−0.13482E−03	−0.17217E−05	−0.26018E−03	−0.64681E−05	0.23215E+04	−0.45441E+03	−0.54210E−19	−0.41923E+01	−0.52241E−01	1
2	0.14350E+05	−0.48970E+04	0.14350E+05	−0.64187E+04	0.73000E+04	−0.36004E+04	0.73000E+04	−0.38956E+04	−0.63757E−05	−0.33742E−04	−0.19765E−04	−0.82464E−04	0.23215E+04	−0.21081E+03	−0.54210E−19	−0.71147E+01	−0.18102E+00	2
3	0.14350E+05	−0.99613E+04	0.14350E+05	−0.38489E+04	0.73000E+04	−0.54430E+03	0.73000E+04	−0.19786E+03	−0.55520E−05	−0.21037E−05	−0.17364E−05	−0.70693E−06	0.23215E+04	−0.22859E+03	−0.54210E−19	−0.45357E+01	−0.51167E−01	3
4	0.14350E+05	−0.63859E+04	0.14350E+05	−0.86239E+04	0.73000E+04	−0.82915E+03	0.73000E+04	−0.32161E+02	−0.31260E−05	−0.42479E−05	−0.23399E−05	−0.20004E−06	0.23215E+04	−0.11524E+03	−0.54210E−19	−0.65518E+01	−0.17332E+00	4
5	0.14350E+05	−0.92977E+04	0.14350E+05	−0.32081E+04	0.73000E+04	−0.10600E+04	0.73000E+04	−0.71121E+03	−0.38437E−04	−0.17491E−05	−0.33550E−05	−0.22127E−05	0.23215E+04	−0.43208E+03	−0.54210E−19	−0.45784E+01	−0.44724E−01	5
6	0.14350E+05	−0.48776E+04	0.14350E+05	−0.64323E+04	0.73000E+04	−0.13162E+04	0.73000E+04	−0.11605E+04	−0.23957E−05	−0.21299E−04	−0.36166E−05	−0.32207E−05	0.23215E+04	−0.20484E+03	−0.54210E−19	−0.70326E+01	−0.16950E+00	6
7	0.14350E+05	−0.64321E+04	0.14350E+05	−0.30769E+04	0.73000E+04	−0.45325E+04	0.73000E+04	−0.61156E+03	−0.15126E−03	−0.17038E−05	−0.21656E−02	−0.20278E−05	0.23215E+04	−0.23016E+03	−0.54210E−19	−0.44018E+01	0.89955E+02	7
8	0.14350E+05	−0.55802E+04	0.14350E+05	−0.64657E+04	0.73000E+04	−0.51804E+03	0.73000E+04	−0.72096E+02	−0.27337E−05	−0.31797E−05	−0.15187E−05	−0.34498E−06	0.23215E+04	−0.11617E+03	−0.54210E−19	−0.72947E+01	0.89831E+02	8
9	0.14350E+05	−0.54825E+04	0.14350E+05	−0.20147E+04	0.73000E+04	−0.54057E+04	0.73000E+04	−0.70954E+03	−0.41879E−03	−0.11055E−05	−0.30598E−02	−0.23805E−05	0.23215E+04	−0.38150E+03	−0.54210E−19	−0.46149E+01	0.89954E+02	9
10	0.14350E+05	−0.40040E+04	0.14350E+05	−0.45520E+04	0.73000E+04	−0.48959E+03	0.73000E+04	−0.67633E+02	−0.19881E−05	−0.22486E−05	−0.14343E−05	−0.30958E−06	0.23215E+04	−0.17498E+03	−0.54210E−19	−0.68775E+01	0.89837E+02	10
11	0.14350E+05	−0.21377E+04	0.14350E+05	−0.10664E+04	0.73000E+04	−0.59253E+04	0.73000E+04	−0.61486E+03	−0.12112E−05	−0.61686E−06	−0.33498E−05	−0.20538E−05	0.23215E+04	−0.23021E+03	−0.54210E−19	−0.42526E+01	0.89954E+02	11
12	0.14350E+05	−0.32725E+04	0.14350E+05	−0.19849E+04	0.73000E+04	−0.54027E+03	0.73000E+04	−0.71120E+02	−0.16199E−05	−0.98677E−06	−0.15852E−05	−0.33624E−06	0.23215E+04	−0.11432E+03	−0.54210E−19	−0.72493E+01	0.89828E+02	12
13	0.14350E+05	−0.19416E+04	0.14350E+05	−0.70423E+03	0.73000E+04	−0.69701E+04	0.73000E+04	−0.67888E+03	−0.11211E−05	−0.41656E−06	−0.49221E−03	−0.23053E−05	0.23215E+04	−0.34956E+03	−0.54210E−19	−0.42798E+01	0.89954E+02	13
14	0.14350E+05	−0.20828E+04	0.14350E+05	−0.13427E+04	0.73000E+04	−0.53654E+03	0.73000E+04	−0.61603E+02	−0.10470E−05	−0.69167E−06	−0.15515E−05	−0.30861E−06	0.23215E+04	−0.16735E+03	−0.54210E−19	−0.68412E+01	0.89832E+02	14
15	0.14350E+05	−0.11396E+04	0.14350E+05	−0.38626E+03	0.73000E+04	−0.58085E+04	0.73000E+04	−0.52003E+03	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.23215E+04	−0.22739E+03	−0.16263E−18	−0.18631E+01	0.89955E+02	15
16	0.14350E+05	−0.11990E+04	0.14350E+05	−0.57206E+00	0.73000E+04	−0.46496E+03	0.73000E+04	−0.21385E−01	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.23215E+04	−0.11448E+03	−0.67763E−19	−0.24499E−01	0.89911E+02	16
17	0.14350E+05	−0.11360E+04	0.14350E+05	−0.38421E+03	0.73000E+04	−0.72736E+04	0.73000E+04	−0.42956E+03	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.23215E+04	−0.22694E+03	−0.16263E−18	−0.25839E+01	0.89935E+02	17
18	0.14350E+05	−0.11779E+04	0.14350E+05	−0.10597E+01	0.73000E+04	−0.52535E+03	0.73000E+04	−0.34101E−01	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.23215E+04	−0.11428E+03	−0.54210E−19	−0.50669E−01	0.89872E+02	18
19	0.18500E+05	−0.96991E+04	0.18500E+05	−0.59088E+04	0.40750E+04	−0.25127E+02	0.40750E+04	−0.11184E+02	−0.50011E−05	−0.30118E−05	−0.79538E−06	−0.65436E−06	0.19250E+04	−0.23215E+01	−0.54210E−19	−0.49008E+00	−0.23310E+00	19
20	0.18500E+05	−0.73713E+04	0.18500E+05	−0.47537E+04	0.40750E+04	−0.39682E+02	0.40750E+04	−0.18827E+02	−0.37941E−05	−0.24282E−05	−0.91626E−06	−0.63890E−06	0.19250E+04	−0.14130E+02	−0.54210E−19	−0.35310E+00	−0.15602E+00	20
21	0.18500E+05	−0.59581E+04	0.18500E+05	−0.97489E+04	0.40750E+04	−0.10285E+02	0.40750E+04	−0.56463E+01	−0.30463E−05	−0.50269E−05	−0.61042E−06	−0.55494E−06	0.19250E+04	−0.15340E+01	−0.54210E−19	−0.24989E+01	−0.10854E+00	21
22	0.18500E+05	−0.47970E+04	0.18500E+05	−0.73832E+04	0.40750E+04	−0.24104E+01	0.40750E+04	−0.41189E+01	−0.24459E−05	−0.38170E−05	−0.56390E−06	−0.57991E−06	0.19250E+04	−0.10424E+02	−0.54210E−19	−0.20028E+01	−0.63203E−01	22
23	0.18500E+05	−0.64254E+04	0.18500E+05	−0.46392E+04	0.40750E+04	−0.37314E+02	0.40750E+04	−0.16479E+02	−0.33063E−05	−0.23629E−05	−0.76136E−06	−0.56449E−06	0.19250E+04	−0.93093E+01	−0.54210E−19	−0.47469E+01	0.58736E+00	23
24	0.18500E+05	−0.45004E+04	0.18500E+05	−0.33770E+04	0.40750E+04	−0.34809E+02	0.40750E+04	−0.17415E+02	−0.23275E−05	−0.17340E−05	−0.84216E−06	−0.65124E−06	0.19250E+04	−0.51787E+01	−0.54210E−19	−0.53250E+01	0.52024E+00	24
25	0.18500E+05	−0.49057E+04	0.18500E+05	−0.66572E+04	0.40750E+04	−0.12972E+01	0.40750E+04	−0.48873E+01	−0.25192E−05	−0.34168E−05	−0.55281E−06	−0.53878E−06	0.19250E+04	−0.73627E+01	−0.54210E−19	−0.24536E+01	0.93331E+00	25
26	0.18500E+05	−0.39276E+04	0.18500E+05	−0.52620E+04	0.40750E+04	−0.58036E+01	0.40750E+04	−0.36755E+01	−0.20215E−05	−0.27119E−05	−0.56283E−06	−0.53700E−06	0.19250E+04	−0.15553E+02	−0.54210E−19	−0.28049E+01	0.91083E+00	26
27	0.18500E+05	−0.20446E+04	0.18500E+05	−0.15287E+04	0.40750E+04	−0.14515E+01	0.40750E+04	−0.13340E+01	−0.10798E−05	−0.80881E−06	−0.55171E−06	−0.55428E−06	0.19250E+04	−0.11554E+02	−0.54210E−19	−0.49770E+01	0.60828E+00	27
28	0.18500E+05	−0.13406E+04	0.18500E+05	−0.10260E+04	0.40750E+04	−0.66623E+01	0.40750E+04	−0.85211E+01	−0.71237E−06	−0.54856E−06	−0.56728E−06	−0.55966E−06	0.19250E+04	−0.45967E+01	−0.54210E−19	−0.50258E+01	0.51836E+00	28
29	0.18500E+05	−0.15994E+04	0.18500E+05	−0.21220E+04	0.40750E+04	−0.98222E+01	0.40750E+04	−0.10461E+02	−0.83785E−06	−0.11176E−05	−0.54382E−06	−0.54848E−06	0.19250E+04	−0.50550E+01	−0.54210E−19	−0.25082E+01	0.96794E+00	29
30	0.18500E+05	−0.14033E+04	0.18500E+05	−0.16503E+04	0.40750E+04	−0.10741E+02	0.40750E+04	−0.19800E+02	−0.73620E−06	−0.86472E−06	−0.57393E−06	−0.68470E−06	0.19250E+04	−0.10281E+02	−0.54210E−19	−0.28742E+01	0.89798E+00	30

NODE	NPZZORY	MY	M	PGAM
10	1	0.63164E+04	−0.79730E+04	−0.14970E−02
10	2	0.20966E+05	−0.76138E+04	−0.67763E−20
11	1	0.63164E+04	−0.97539E+04	−0.47667E−02
12	1	0.63164E+04	−0.80533E+04	−0.16704E−02
12	2	0.20966E+05	−0.65492E+04	−0.54210E−19
13	1	0.63164E+04	−0.86727E+04	−0.32142E−02
15	1	0.63164E+04	−0.80253E+04	−0.18633E−02
16	1	0.63164E+04	−0.72405E+04	−0.54228E−03
18	1	0.63164E+04	−0.63015E+04	−0.67763E−20
19	1	0.63164E+04	−0.70048E+04	−0.30069E−03
19	2	0.20966E+05	−0.49822E+04	−0.54210E−19
20	1	0.63164E+04	−0.86822E+04	−0.24966E−02
21	1	0.63164E+04	−0.72503E+04	−0.55391E−03
21	2	0.20966E+05	−0.42740E+04	−0.23717E−19
22	1	0.63164E+04	−0.77730E+04	−0.13513E−02
24	1	0.63164E+04	−0.68074E+04	−0.15288E−03
25	1	0.63164E+04	−0.64207E+04	−0.68945E−05
27	1	0.63164E+04	−0.49879E+04	−0.65052E−18

Figure 7.21: 2-Story Building: Output file, $for006$ (Contd.)

MAXIMUM STATIC PLUS DYNAMIC ELEMENT RESPONSES

DATE 06/20/2003 TIME 19:51:56

ELEM	MPYL1	MYL1	MPYL2	MYL2	MPZL1	MZL1	MPZL2	MZL2	KAPYL1	KAPYL2	KAPZL1	KAPZL2	PY	P	PEPS	TORSION,MXL		ALPHA	ELEM
1	0.14350E+05	0.10053E+05	0.14350E+05	0.36773E+04	0.73000E+04	0.39519E+04	0.73000E+04	0.31300E+04	0.17223E-02	0.20025E-05	0.24632E-02	0.95280E-05	0.23215E+04	0.97137E+02	0.13553E-19	0.38680E+01	-0.52241E-01		1
2	0.14350E+05	0.56184E+04	0.14350E+05	0.76542E+04	0.73000E+04	0.50759E+04	0.73000E+04	0.53685E+04	0.11106E-03	0.72711E-03	0.25420E-03	0.13065E-02	0.23215E+04	0.28470E+02	0.13553E-19	0.64600E+01	-0.18102E+00		2
3	0.14350E+05	0.12461E+05	0.14350E+05	0.49841E+04	0.73000E+04	0.25613E+04	0.73000E+04	0.57138E+03	0.60915E-03	0.27354E-05	0.88649E-03	0.17340E-05	0.23215E+04	-0.17824E+03	0.13553E-19	0.38096E+01	-0.51167E-01		3
4	0.14350E+05	0.74110E+04	0.14350E+05	0.10372E+05	0.73000E+04	0.24275E+03	0.73000E+04	0.97631E+02	0.36047E-05	0.50855E-05	0.74010E-06	0.42500E-06	0.23215E+04	-0.87920E+02	0.13553E-19	0.60257E+01	-0.17332E+00		4
5	0.14350E+05	0.12094E+05	0.14350E+05	0.38870E+04	0.73000E+04	0.48142E+04	0.73000E+04	0.34117E+04	0.28195E-03	0.20978E-05	0.40188E-03	0.10365E-04	0.23215E+04	0.51953E+02	0.13553E-19	0.33457E+01	-0.44724E-01		5
6	0.14350E+05	0.54422E+04	0.14350E+05	0.80921E+04	0.73000E+04	0.40413E+04	0.73000E+04	0.46669E+04	0.26554E-05	0.81032E-06	0.10885E-04	0.59688E-04	0.23215E+04	0.11648E+02	0.13553E-19	0.67526E+01	-0.16950E+00		6
7	0.14350E+05	0.89847E+04	0.14350E+05	0.37899E+04	0.73000E+04	0.46178E+04	0.73000E+04	0.28686E+03	0.13532E-02	0.21031E-05	0.32699E-03	0.11787E-05	0.23215E+04	-0.17359E+03	0.13553E-19	0.33212E+01	0.89955E+02		7
8	0.14350E+05	0.66237E+04	0.14350E+05	0.88996E+04	0.73000E+04	0.63892E+03	0.73000E+04	0.54322E+02	0.32187E-05	0.43586E-05	0.18478E-05	0.28969E-06	0.23215E+04	-0.80857E+02	0.13553E-19	0.67630E+01	0.89831E+02		8
9	0.14350E+05	0.79253E+04	0.14350E+05	0.28410E+04	0.73000E+04	0.59498E+04	0.73000E+04	0.64903E+03	0.17493E-02	0.15588E-05	0.99177E-03	0.22661E-05	0.23215E+04	-0.76960E+02	0.13553E-19	0.33977E+01	0.89954E+02		9
10	0.14350E+05	0.51310E+04	0.14350E+05	0.69754E+04	0.73000E+04	0.60201E+03	0.73000E+04	0.53271E+02	0.25173E-05	0.34288E-05	0.17083E-05	0.26635E-06	0.23215E+04	-0.44984E+02	0.13553E-19	0.60999E+01	0.89837E+02		10
11	0.14350E+05	0.94356E+04	0.14350E+05	0.45333E+04	0.73000E+04	0.47358E+04	0.73000E+04	0.22014E+03	0.13014E-03	0.24846E-05	0.27128E-03	0.78639E-06	0.23215E+04	-0.17711E+03	0.13553E-19	0.34380E+01	0.89954E+02		11
12	0.14350E+05	0.54159E+04	0.14350E+05	0.76262E+04	0.73000E+04	0.64063E+03	0.73000E+04	0.68093E+02	0.26402E-05	0.37504E-05	0.18436E-05	0.32559E-06	0.23215E+04	-0.86077E+02	0.13553E-19	0.68274E+01	0.89828E+02		12
13	0.14350E+05	0.83009E+04	0.14350E+05	0.37928E+04	0.73000E+04	0.61259E+04	0.73000E+04	0.42016E+03	0.46650E-03	0.20878E-05	0.94692E-03	0.17029E-05	0.23215E+04	-0.15389E+03	0.13553E-19	0.34983E+01	0.89954E+02		13
14	0.14350E+05	0.38627E+04	0.14350E+05	0.54155E+04	0.73000E+04	0.57111E+03	0.73000E+04	0.67379E+02	0.18874E-05	0.26798E-05	0.16237E-05	0.31270E-06	0.23215E+04	-0.78850E+02	0.13553E-19	0.63865E+01	0.89832E+02		14
15	0.14350E+05	0.55253E+04	0.14350E+05	0.11994E+04	0.73000E+04	0.47522E+04	0.73000E+04	0.46505E+03	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.23215E+04	-0.17814E+03	0.54210E-19	0.30094E+01	0.89955E+02		15
16	0.14350E+05	0.38626E+03	0.14350E+05	0.18241E+01	0.73000E+04	0.51921E+03	0.73000E+04	0.22833E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.23215E+04	-0.87783E+02	0.40658E-19	0.15838E-01	0.89911E+02		16
17	0.14350E+05	0.55011E+04	0.14350E+05	0.11781E+04	0.73000E+04	0.64769E+04	0.73000E+04	0.52555E+03	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.23215E+04	-0.17814E+03	0.21684E-18	0.43532E+01	0.89935E+02		17
18	0.14350E+05	0.38421E+03	0.14350E+05	0.29037E+01	0.73000E+04	0.42824E+03	0.73000E+04	0.33816E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.23215E+04	-0.87287E+02	0.13553E-19	0.44543E-01	0.89872E+02		18
19	0.18500E+05	0.11535E+05	0.18500E+05	0.70743E+04	0.40750E+04	0.44898E+01	0.40750E+04	0.36133E+01	0.59328E-05	0.35720E-05	0.54973E-06	0.56016E-06	0.19250E+04	0.11702E+02	0.13553E-19	0.21012E+01	-0.23310E+00		19
20	0.18500E+05	0.88716E+04	0.18500E+05	0.55863E+04	0.40750E+04	0.51844E+01	0.40750E+04	0.18591E+01	0.45698E-05	0.28353E-05	0.57265E-06	0.57199E-06	0.19250E+04	0.90702E+01	0.13553E-19	0.14629E+01	-0.15602E+00		20
21	0.18500E+05	0.73213E+04	0.18500E+05	0.11875E+05	0.40750E+04	0.11235E+02	0.40750E+04	0.15328E+02	0.37049E-05	0.60968E-05	0.55776E-06	0.59651E-06	0.19250E+04	0.60535E+01	0.13553E-19	0.15844E+01	-0.10854E+00		21
22	0.18500E+05	0.58530E+04	0.18500E+05	0.92055E+04	0.40750E+04	0.11788E+02	0.40750E+04	0.26144E+02	0.29797E-05	0.47395E-05	0.60610E-06	0.82106E-06	0.19250E+04	0.17574E+02	0.13553E-19	0.11804E+01	-0.63203E-01		22
23	0.18500E+05	0.97858E+04	0.18500E+05	0.59787E+04	0.40750E+04	0.34933E+01	0.40750E+04	0.31647E+01	0.50422E-05	0.30403E-05	0.55922E-06	0.56235E-06	0.19250E+04	0.41717E+01	0.13553E-19	0.44123E+01	0.58736E+00		23
24	0.18500E+05	0.63356E+04	0.18500E+05	0.44928E+04	0.40750E+04	0.13422E+02	0.40750E+04	0.11422E+02	0.32646E-05	0.23007E-05	0.56169E-06	0.55385E-06	0.19250E+04	0.17406E+02	0.13553E-19	0.47162E+01	0.52024E+00		24
25	0.18500E+05	0.63407E+04	0.18500E+05	0.10117E+05	0.40750E+04	0.16225E+02	0.40750E+04	0.14659E+02	0.32338E-05	0.52012E-05	0.56102E-06	0.59960E-06	0.19250E+04	0.96686E+01	0.13553E-19	0.28562E+01	0.93331E+00		25
26	0.18500E+05	0.54136E+04	0.18500E+05	0.79767E+04	0.40750E+04	0.16365E+02	0.40750E+04	0.24337E+02	0.27684E-05	0.41005E-05	0.60511E-06	0.70018E-06	0.19250E+04	0.20924E+02	0.13553E-19	0.29613E+01	0.91083E+00		26
27	0.18500E+05	0.85375E+04	0.18500E+05	0.54960E+04	0.40750E+04	0.29202E+02	0.40750E+04	0.14246E+02	0.43899E-05	0.27959E-05	0.81157E-06	0.64469E-06	0.19250E+04	0.42080E+01	0.13553E-19	0.44317E+01	0.60828E+00		27
28	0.18500E+05	0.53891E+04	0.18500E+05	0.40261E+04	0.40750E+04	0.27290E+02	0.40750E+04	0.14752E+02	0.27799E-05	0.20575E-05	0.67597E-06	0.60868E-06	0.19250E+04	0.18356E+02	0.13553E-19	0.46760E+01	0.51836E+00		28
29	0.18500E+05	0.56584E+04	0.18500E+05	0.87169E+04	0.40750E+04	0.66680E+00	0.40750E+04	0.44350E+00	0.28867E-05	0.45015E-05	0.37894E-06	0.25624E-06	0.19250E+04	0.50121E+01	0.13553E-19	0.28665E+01	0.96794E+00		29
30	0.18500E+05	0.44487E+04	0.18500E+05	0.61652E+04	0.40750E+04	0.37575E+01	0.40750E+04	0.57990E+01	0.22831E-05	0.31736E-05	0.57052E-06	0.53176E-06	0.19250E+04	0.21353E+02	0.13553E-19	0.29562E+01	0.89798E+00		30

NODE	NPZZORY	MY	M	PGAM
10	1	0.63164E+04	0.69481E+04	0.25314E-03
10	2	0.20966E+05	0.49500E+04	0.43368E-18
11	1	0.63164E+04	0.84475E+04	0.29072E-02
12	1	0.63164E+04	0.69501E+04	0.25470E-03
12	2	0.20966E+05	0.15700E+04	0.10842E-18
13	1	0.63164E+04	0.70559E+04	0.34704E-03
15	1	0.63164E+04	0.24807E+04	0.10842E-18
16	1	0.63164E+04	0.48378E+04	0.21684E-18
18	1	0.63164E+04	0.15285E+04	0.19516E-17
19	1	0.63164E+04	0.59356E+04	0.21684E-18
19	2	0.20966E+05	0.35540E+04	0.10842E-18
20	1	0.63164E+04	0.76275E+04	0.10939E-02
21	1	0.63164E+04	0.59399E+04	0.24395E-18
21	2	0.20966E+05	0.10607E+04	0.31171E-18
22	1	0.63164E+04	0.57743E+04	0.13010E-17
24	1	0.63164E+04	0.17674E+04	0.65052E-18
25	1	0.63164E+04	0.41799E+04	0.43368E-18
27	1	0.63164E+04	0.12340E+04	0.27105E-19

Figure 7.22: 2-Story Building: Output file, $for006$ (Contd.)

DATE 06/20/2003 TIME 19:51:56

PANEL ZONE INELASTIC ROTATIONS

NODE	YRPZ	ZRPZ
1	0.00	0.00
2	0.00	0.00
3	0.00	0.00
4	0.00	0.00
5	0.00	0.00
6	0.00	0.00
7	0.00	0.00
8	0.00	0.00
9	0.00	0.00
10	0.00	0.15
11	0.00	0.48
12	0.00	0.17
13	0.00	0.32
14	0.00	0.00
15	0.00	0.19
16	0.00	0.05
17	0.00	0.00
18	0.00	0.00
19	0.00	0.03
20	0.00	0.25
21	0.00	0.06
22	0.00	0.14
23	0.00	0.00
24	0.00	0.02
25	0.00	0.00
26	0.00	0.00
27	0.00	0.00

BEAM/COLUMN END INELASTIC ROTATIONS

ELEM	N1	N1ATTACH	YROT1	ZROT1	N2	N2ATTACH	YROT2	ZROT2
1	1	0	0.17	0.25	10	6	0.00	0.00
2	10	5	0.01	0.03	19	6	0.07	0.13
3	2	0	0.06	0.09	11	6	0.00	0.00
4	11	5	0.00	0.00	20	6	0.00	0.00
5	3	0	0.03	0.04	12	6	0.00	0.00
6	12	5	0.00	0.00	21	6	0.00	0.01
7	4	0	0.14	0.22	13	6	0.00	0.00
8	13	5	0.00	0.00	22	6	0.00	0.00
9	7	0	0.17	0.31	16	6	0.00	0.00
10	16	5	0.00	0.00	25	6	0.00	0.00
11	6	0	0.01	0.03	15	6	0.00	0.00
12	15	5	0.00	0.00	24	6	0.00	0.00
13	9	0	0.05	0.09	18	6	0.00	0.00
14	18	5	0.00	0.00	27	6	0.00	0.00
15	5	0	0.00	0.00	14	0	0.00	0.00
16	14	0	0.00	0.00	23	0	0.00	0.00
17	8	0	0.00	0.00	17	0	0.00	0.00
18	17	0	0.00	0.00	26	0	0.00	0.00
19	10	1	0.00	0.00	11	2	0.00	0.00
20	19	1	0.00	0.00	20	2	0.00	0.00
21	11	1	0.00	0.00	12	2	0.00	0.00
22	20	1	0.00	0.00	21	2	0.00	0.00
23	10	4	0.00	0.00	13	2	0.00	0.00
24	19	4	0.00	0.00	22	2	0.00	0.00
25	13	1	0.00	0.00	16	2	0.00	0.00
26	22	1	0.00	0.00	25	2	0.00	0.00
27	12	4	0.00	0.00	15	2	0.00	0.00
28	21	4	0.00	0.00	24	2	0.00	0.00
29	15	1	0.00	0.00	18	2	0.00	0.00
30	24	1	0.00	0.00	27	2	0.00	0.00

PRINCIPAL STRESSES AND MAXIMUM SHEAR STRESS IN PLANE STRESS ELEMENTS

ELEM. NO.	MAX. SIGMA1	ANGLE	MIN. SIGMA2	ANGLE	MAX. SHEAR STRESS
1	0.12679886	45.99663587	−0.15288483	45.99663587	0.13984185
2	0.09927425	−44.89577728	−0.10511986	−46.39223013	0.10218357
3	0.07395474	38.57957058	−0.10974282	38.57957058	0.09184878
4	0.07023142	−48.03172168	−0.09353122	34.99353682	0.06192345
5	0.23955270	−45.83219764	−0.31261173	−45.83219764	0.27608222
6	0.20445100	44.90472800	−0.22223180	44.90911577	0.21329384
7	0.16577347	47.38731017	−0.19988095	−50.66346429	0.16959102
8	0.13964590	−54.35553228	−0.16048637	44.02047764	0.13520314

Figure 7.23: 2-Story Building: Output file, $for006$ (Contd.)

Appendix A Glossary of Input & Output Variables

Variable	I/O File	Data Block	Description
$A0$	$for005$	Integration & Iteration Data	Rayleigh damping parameter.
$A1$	$for005$	Integration & Iteration Data	Rayleigh damping parameter.
$AGRAV$	$for005$	Integration & Iteration Data	Acceleration due to gravity.
$ALPHA$	$for005$	Beam Element Data	Element orientation angle.
$ALPHA$	$for006$	Static and Dynamic Element Responses	Updated orientation angle of the element.
$AREA$	$for009$	Beam Section Data	Area of the cross-section.
$BETA$	$for005$	Integration & Iteration Data	Newmark integration parameter.
DAM_IND	$for006$,	Building Damage Indices	Damage index of elastofiber/MEF and type 1 panel zone elements.
$DEPTH$	$for009$	Beam Section Data	Section depth.
DOF	$for005$	Spring Element Data	The degree-of-freedom (1-8) the motion of which is resisted by the spring.
DT	$for005$	Integration & Iteration Data	Time increment for dynamic time-history analysis.
$DTHK$	$for005$	Panel Zone Element Data	Doubler plate thickness.
$DTOUT$	$for005$	Deformed Shape Output Control Data	Time interval at which deformed shape nodal coordinates are required.
E	$for005$	Diaphragm Element Material Data	Elastic modulus.
$ECCMIN$	$for005$	Beam Element Data	Member initial imperfection in minor direction, along Y' axis.
$ECCMAJ$	$for005$	Beam Element Data	Member initial imperfection in major direction, along Z' axis.
EE	$for005$	Beam Element Data	Modulus of Elasticity.
$ELEM$	$for005$	Time-History Output Control Data	Beam element number.
$ELEM$	$for005$	Time-History Output Control Data	Elastofiber type beam element number.
$ELEM$	$for006$	Static and Dynamic Element Responses	Beam element number.
$ELEM$	$for006$	Static Element Responses	Beam element number.
$EPSS$	$for005$	Beam Element Data	Fiber strain at onset of strain-hardening in an elastofiber element.
$EPSU$	$for005$	Beam Element Data	Fiber ultimate strain in an elastofiber element.
ES	$for005$	Beam Element Data	Fiber tangent modulus (slope of stress-strain curve) in an elastofiber element at onset of strain-hardening.
$FAC1$	$for005$	Beam Element Data	Degree of continuity at end i.
$FAC2$	$for005$	Beam Element Data	Degree of continuity at end j.
$FAFRAC$	$for005$	Categories for Elastofiber Element Fiber Area Reduction	Fraction of fiber area to be used in the analysis (0.0-1.0).
FIB	$for005$	Time-History Output Control Data	Fiber number.
$FLGWDTH$	$for009$	Beam Section Data	Flange width.
$FLGTHK$	$for009$	Beam Section Data	Flange thickness.

Variable	I/O File	Data Block	Description
FX	$for005$	Nodal Data	Static nodal force in the global X direction.
FY	$for005$	Nodal Data	Static nodal force in the global Y direction.
$FYFRAC(10)$	$for005$	Categories for Elastofiber Element Fiber Fracture Stress Prob. Distribution	Fracture strain probability distribution.
FZ	$for005$	Nodal Data	Static nodal force in the global Z direction.
G	$for005$	Panel Zone Element Data	Shear modulus.
$GAMMA$	$for005$	Integration & Iteration Data	Newmark integration parameter.
$GAMULTX$	$for005$	Ground Acceleration Scaling Factor	Scaling factor for X-component of ground acceleration.
$GAMULTY$	$for005$	Ground Acceleration Scaling Factor	Scaling factor for Y-component of ground acceleration.
$GAMULTZ$	$for005$	Ground Acceleration Scaling Factor	Scaling factor for Z-component of ground acceleration.
HPZ	$for005$	Panel Zone Element Data	Panel zone height.
HSH	$for005$	Beam Element Data	Strain hardening modulus.
HSH	$for005$	Panel Zone Element Data	Strain hardening modulus.
IBC	$for005$	Beam Element Data	Beam or column identifier: 1 for Beam 2 for Column.
$ICAT$	$for005$	Categories for Elastofiber Element Fiber Area Reduction	Fiber area reduction category number.
$ICAT$	$for005$	Categories for Elastofiber Element Fiber Fracture Stress Prob. Distribution	Fiber fracture strain probability distribution category number.
$ICATFAFRAC$	$for005$	Beam Element Data	Category for fiber area reduction.
$ICATFYFRAC$	$for005$	Beam Element Data	Category for fiber fracture strain.
$IDASSCOL$	$for005$	Nodal Data	Element number of the column that is to be associated with the joint under consideration.
$IDES$	$for005$	Beam Element Data	Section designation.
$IDES$	$for009$	Beam Section Data	Section designation (label).
$IDFIB(20)$	$for005$	Categories for Elastofiber Element Fiber Area Reduction	Fiber numbers.
$IDFIB(20)$	$for005$	Categories for Elastofiber Element Fiber Fracture Strain Prob. Distribution	Fiber numbers.
$IDSEG$	$for005$	Categories for Elastofiber Element Fiber Area Reduction	Fiber segment 1 or 2.
$IDSEG$	$for005$	Categories for Elastofiber Element Fiber Fracture Strain Prob. Distribution	Fiber segment 1 or 2.
$IELTYPE$	$for005$	Beam Element Data	Element type: 0 for plastic-hinge, 1 for elastofiber.
$IPZTYPE$	$for005$	Panel Zone Element Data	Panel zone type: 0 for bilinear and 1 for linear-quadratic.
$ISEED$	$for005$	Seed for Random Number Generation for Elastofiber Element Fiber Fracture Strain Computation	Seed to generate random numbers.

Variable	I/O File	Data Block	Description
$ITHR$	$for005$	Integration & Iteration Data	Threshold DOF for system picture.
$KAPYL1$	$for006$	Static and Dynamic Element Responses	Kink rotation in the element at node 1 about the local Y' (major) axis.
$KAPYL2$	$for006$	Static and Dynamic Element Responses	Kink rotation in the element at node 2 about the local Y' (major) axis.
$KAPZL1$	$for006$	Static and Dynamic Element Responses	Kink rotation in the element at node 1 about the local Z' (minor) axis.
$KAPZL2$	$for006$	Static and Dynamic Element Responses	Kink rotation in the element at node 2 about the local Z' (minor) axis.
$LENGTH$	$for006$	Beam Element Data	Clear length of the beam element between attachment points.
LMG	$for005$	Diaphragm Element Connectivity Data	Increment to be applied to the node numbers for the next element in the series to be generated.
M	$for006$	Static and Dynamic Element Responses	Panel zone moment.
MAT	$for005$	Diaphragm Element Connectivity Data	Material number.
MBD	$for005$	Main Control Data	Global half bandwidth.
$MPYL1$	$for006$	Static and Dynamic Element Responses	Plastic moment capacity at node 1 of the element about the local Y' (major) axis.
$MPYL2$	$for006$	Static and Dynamic Element Responses	Plastic moment capacity at node 2 of the element about the local Y' (major) axis.
$MPZL1$	$for006$	Static and Dynamic Element Responses	Plastic moment capacity at node 1 of the element about the local Z' (minor) axis.
$MPZL2$	$for006$	Static and Dynamic Element Responses	Plastic moment capacity at node 2 of the element about the local Z' (minor) axis.
MTP	$for005$	Main Control Data	Maximum number of turning points.
MX	$for005$	Nodal Data	Nodal mass in weight units in the global X direction.
MXL	$for006$	Static and Dynamic Element Responses	Torsion in the element.
MY	$for005$	Nodal Data	Nodal mass in weight units in the global Y direction.
MY	$for006$	Static and Dynamic Element Responses	Panel zone yield moment.
$MYL1$	$for006$	Static and Dynamic Element Responses	Moment in the element at node 1 about the local Y' (major) axis.
$MYL2$	$for006$	Static and Dynamic Element Responses	Moment in the element at node 2 about the local Y' (major) axis.
MZ	$for005$	Nodal Data	Nodal mass in weight units in the global Z direction.
$MZL1$	$for006$	Static and Dynamic Element Responses	Moment in the element at node 1 about the local Z' (minor) axis.
$MZL2$	$for006$	Static and Dynamic Element Responses	Moment in the element at node 2 about the local Z' (minor) axis.
N	$for005$	Diaphragm Element Connectivity Data	First element in the series to be generated.
$N1$	$for005$	Diaphragm Element Connectivity Data	Node 1 of first element in the series to be generated.
$N1$	$for005$	Diaphragm Element Output Control Data	First diaphragm element in the series for which principal stress-strain histories are required.
$N1ATTACH$	$for006$	Beam Element Data	Attachment point of element at node 1.
$N2$	$for005$	Diaphragm Element	Node 2 of first element in the series

Variable	I/O File	Data Block	Description
		Connectivity Data	to be generated.
$N2$	$for005$	Diaphragm Element Output Control Data	Last diaphragm element in the series for which principal stress-strain histories are required.
$N2ATTACH$	$for006$	Beam Element Data	Attachment point of element at node 2.
$N3$	$for005$	Diaphragm Element Connectivity Data	Node 3 of first element in the series to be generated.
$N4$	$for005$	Diaphragm Element Connectivity Data	Node 4 of first element in the series to be generated.
$NCURVES$	$for005$	Main Control Data	Number of curves in the PMM interaction surface.
$NDIM$	$for005$	Main Control Data	Storage parameter for turning point locations.
$NDRIFTX$	$for005$	Interstory Drift Output Control Data	Number of story drifts in the X direction to be output
$NDRIFTY$	$for005$	Interstory Drift Output Control Data	Number of story drifts in the Y direction to be output
$NDSTPS$	$for005$	Main Control Data	Number of dynamic analysis steps.
NE	$for005$	Diaphragm Element Connectivity Data	Last element in the series to be generated.
$NEIG$	$for005$	Main Control Data	Number of eigen values and vectors to be computed.
$NELMGRPMAX$	$for005$	Main Control Data	Maximum number of elements in any of the $NGRP$ groups.
$NFIBEL$	$for005$	Main Control Data	Number of elastofiber beam elements.
$NFIBEL5$	$for005$	Main Control Data	Number of 5-segment modified elastofiber (MEF) elements.
NG	$for005$	Diaphragm Element Connectivity Data	Increment in diaphragm element number.
$NGITMAX$	$for005$	Main Control Data	Maximum number of global iterations per time step.
$NGRP$	$for005$	Main Control Data	Number of element groups for which cumulative resultant forces are to be output.
$NINTPTPLSTR$	$for005$	Diaphragm Element Control Data	Number of integration points for Gauss quadrature.
$NLITMAX$	$for005$	Main Control Data	Maximum number of local member (elastofiber element) iterations per time step.
NNP	$for005$	Main Control Data	Number of node points.
$NODE$	$for005$	Spring Element Data	Identity of the node-point at which the spring is to be located.
$NODE$	$for005$	Panel Zone Element Data	Node number.
$NODE$	$for005$	Time-History Output Control Data	Node number.
$NODE$	$for005$	Time-History Output Control Data	Node number.
$NODE$	$for006$	Static and Dynamic Element Responses	Node number.
$NODE1$	$for005$	Beam Element Data	Node number at end i of the element.
$NODE2$	$for005$	Beam Element Data	Node number at end j of the element.
$NOUT$	$for005$	Main Control Data	Output data is written for every $NOUT^{th}$ time step.
$NPCURVE$	$for005$	Main Control Data	Number of points in each curve comprising the PMM interaction surface.
$NPHEL$	$for005$	Main Control Data	Number of plastic hinge beam elements.

Variable	I/O File	Data Block	Description
$NPLSTRSS$	$for005$	Diaphragm Element Control Data	Number of diaphragm elements for which principal stress-strain history is required.
$NPZ0$	$for005$	Main Control Data	Number of panel zones with bilinear shear stress-strain relation.
$NPZ1$	$for005$	Main Control Data	Number of panel zones with linear-quadratic shear stress-strain relation.
$NPZZORY$	$for005$	Panel Zone Element Data	Panel zone orientation: 1 for $X'Z'$ plane and 2 for $X'Y'$ plane.
$NPZZORY$	$for006$	Static and Dynamic Element Responses	Panel zone orientation: 1 for $X'Z'$ plane and 2 for $X'Y'$ plane.
$NRTH$	$for005$	Main Control Data	Number of response time histories to be output.
$NSPR$	$for005$	Main Control Data	Number of translational and rotational springs.
$NSSTPS$	$for005$	Main Control Data	Number of static analysis steps.
$NTHR$	$for005$	Integration & Iteration Data	Threshold node for system picture.
$NUMPLSTREL$	$for005$	Diaphragm Element Control Data	Number of diaphragm elements.
$NUMPLSTRMAT$	$for005$	Diaphragm Element Control Data	Number of diaphragm element material types.
$OLFSRATIO$	$for005$	Beam Element Data	Ratio of fiber segment length to element total length.
P	$for006$	Static and Dynamic Element Responses	Axial force in the element.
$PEPS$	$for006$	Static and Dynamic Element Responses	Axial strain in the element.
$PGAM$	$for006$	Static and Dynamic Element Responses	Plastic shear strain in the panel zone.
$POIS$	$for005$	Diaphragm Element Material Data	Poisson's ratio.
$PSTRESS$	$for006$	Static Element Responses	Axial stress in the element.
PY	$for006$	Static and Dynamic Element Responses	Axial yield force of the element.
$RESPID$	$for005$	Time-History Output Control Data	Nodal response quantity.
$RESPID$	$for005$	Time-History Output Control Data	Element response quantity.
$RESPID$	$for005$	Time-History Output Control Data	Panel zone response quantity.
$RESPID$	$for005$	Time-History Output Control Data	Fiber response quantity.
$RTH1$	$for005$	Peak Interstory Drift Output Control Data	ID of the time-history that corresponds to the displacement of the bottom of the story.
$RTH2$	$for005$	Peak Interstory Drift Output Control Data	ID of the time-history that corresponds to the displacement of the top of the story.
$RXRES$	$for005$	Nodal Data	Restraint to nodal rotation about joint local $\bar{X}$ axis.
$RYRES$	$for005$	Nodal Data	Restraint to nodal rotation about joint local $\bar{Y}$ axis.
$RZRES$	$for005$	Nodal Data	Restraint to nodal rotation about joint local $\bar{Z}$ axis.
SEG	$for005$	Time-History Output Control Data	Segment number.
$SIGU$	$for005$	Beam Element Data	Fiber ultimate stress in an elastofiber element.

Variable	I/O File	Data Block	Description
$SIGY$	$for005$	Beam Element Data	Yield stress.
SMI	$for009$	Beam Section Data	Moment of inertia about the strong (major) axis of the section.
$SPRSTF$	$for005$	Spring Element Data	Spring stiffness.
SSM	$for009$	Beam Section Data	Section modulus about the strong (major) axis of the section.
$STFFACFE$	$for005$	Integration & Iteration Data	Fraction of elastic stiffness of elastofiber element to be added to its tangent stiffness for Newton-Raphson iteration.
$STFFACPHE$	$for005$	Integration & Iteration Data	Fraction of elastic stiffness of plastic hinge element to be added to its tangent stiffness for Newton-Raphson iteration.
$STFFACPZ0$	$for005$	Integration & Iteration Data	Fraction of elastic stiffness of bilinear panel zone element to be added to its tangent stiffness for Newton-Raphson iteration.
$STFFACPZ1$	$for005$	Integration & Iteration Data	Fraction of elastic stiffness of linear-quadratic panel zone element to be added to its tangent stiffness for Newton-Raphson iteration.
$STORYHT$	$for005$	Peak Interstory Drift Output Control Data	Story height.
SZ	$for009$	Beam Section Data	Plastic section modulus about the strong (major) axis of the section.
$TAUY$	$for005$	Panel Zone Element Data	Shear yield stress.
THK	$for005$	Panel Zone Element Data	Panel zone thickness.
THK	$for005$	Diaphragm Element Connectivity Data	Diaphragm element thickness.
$THRESH$	$for005$	Integration & Iteration Data	Threshold value for system picture.
$TIMEOSTF$	$for005$	Integration & Iteration Data	Time till which the initial elastic global stiffness matrix is to be used for Newton-Raphson iteration.
$TOLF$	$for005$	Integration & Iteration Data	Force tolerance for global convergence of displacement DOF.
$TOLFIBELF$	$for005$	Integration & Iteration Data	Force tolerance for local convergence of displacement DOF of an elastofiber element.
$TOLFIBELM$	$for005$	Integration & Iteration Data	Moment tolerance for local convergence of rotation DOF of an elastofiber element.
$TOLM$	$for005$	Integration & Iteration Data	Moment tolerance for global convergence of rotation DOF.
$TORJ$	$for009$	Beam Section Data	Torsional constant.
$TSTRESS$	$for006$	Static Element Responses	Total stress in the element.
$UNITS$	$for005$	Yield Stress Units	Units used for the yield stress.
$WEBTHK$	$for009$	Beam Section Data	Web thickness.
WMI	$for009$	Beam Section Data	Moment of inertia about the weak (minor) axis of the section.
WSM	$for009$	Beam Section Data	Section modulus about the weak (minor) axis of the section.
WZ	$for009$	Beam Section Data	Plastic section modulus about the weak (minor) axis of the section.
X	$for005$	Nodal Data	X coordinate of node-point.
$XDISP$	$for006$	Static Element Responses	Displacement of the node in the global X direction.
$XRES$	$for005$	Nodal Data	Restraint to nodal translation in global

Variable	I/O File	Data Block	Description
			X direction.
Y	$for005$	Nodal Data	Y coordinate of node-point.
$YDISP$	$for006$	Static Element Responses	Displacement of the node in the global Y direction.
$YLMSTRESS$	$for006$	Static Element Responses	Greater of the flexural stresses at the two ends of the element about the local Y' (major) axis.
$YRES$	$for005$	Nodal Data	Restraint to nodal translation in global Y direction.
$YROT1$	$for006$	Beam Element End Inelastic Rotations	Plastic rotation (%) at node 1 of element about the local Y' (major) axis.
$YROT2$	$for006$	Beam Element End Inelastic Rotations	Plastic rotation (%) at node 2 of element about the local Y' (major) axis.
$YRPZ$	$for006$	Panel Zone Inelastic Rotations	Plastic rotation (%) in a $X'Y'$ panel zone.
Z	$for005$	Nodal Data	Z coordinate of node-point.
$ZDISP$	$for006$	Static Element Responses	Displacement of the node in the global Z direction.
$ZLMSTRESS$	$for006$	Static Element Responses	Greater of the flexural stresses at the two ends of the element about the local Z' (minor) axis.
$ZRES$	$for005$	Nodal Data	Restraint to nodal translation in global Z direction.
$ZROT1$	$for006$	Beam Element End Inelastic Rotations	Plastic rotation (%) at node 1 of element about the local Z' (minor) axis.
$ZROT2$	$for006$	Beam Element End Inelastic Rotations	Plastic rotation (%) at node 2 of element about the local Z' (minor) axis.
$ZRPZ$	$for006$	Panel Zone Inelastic Rotations	Plastic rotation (%) in a $X'Z'$ panel zone.

Bibliography

Bathe, K.-J. (1996). *Finite Element Procedures*. Prentice-Hall of India Private Limited, New Delhi.

Bertero, V. V., E. P. Popov, and H. Krawinkler (1972). Beam-column subassemblages under repeated loading. *Journal of the Structural Division, ASCE 98*(ST5), 1137–1159.

Challa, V. R. M. (1992). Nonlinear seismic behavior of steel planar moment-resisting frames. Technical Report EERL 92-01, Earthquake Engineering Research Laboratory, California Institute of Technology, Pasadena, California.

Cook, R. D., D. S. Malkus, and M. E. Plesha (1989). *Concepts and Applications of Finite Element Analysis (Third Edition)*. John Wiley and Sons, New York.

FEMA (2000). *Prestandard and Commentary for the Seismic Rehabilitation of Buildings*. FEMA-356. Federal Emergency Management Agency, USA.

Fielding, D. J. and W. F. Chen (1973). Steel frame analysis and connection shear deformation. *Journal of the Structural Division, ASCE 99*(ST1), 1–18.

Hall, J. F. and V. R. M. Challa (1995). Beam-column modeling. *Journal of Engineering Mechanics 121*(12), 1284–1291.

ICBO (1997). *1997 Uniform Building Code*. Volume 2. International Conference of Building Officials, Whittier, California, USA.

Kassimali, A. (1983). Large deformation analysis of elastic-plastic frames. *Journal of the Structural Division, ASCE 109*(8), 1869–1886.

Kato, B. (1982). Beam-to-column connection research in japan. *Journal of the Structural Division, ASCE 108*(ST2), 343–360.

Krawinkler, H., V. V. Bertero, and E. P. Popov (1975). Shear behavior of steel frame joints. *Journal of the Structural Division, ASCE 101*(ST11), 2317–2336.

Krawinkler, H. and E. P. Popov (1982). Shear behavior of moment connections and joints. *Journal of the Structural Division, ASCE 108*(ST2), 373–391.

Krishnan, S. (2003). Three-dimensional nonlinear analysis of tall irregular steel buildings subject to strong ground motion. Technical Report EERL 2003-01, Earthquake Engineering Research Laboratory, California Institute of Technology, Pasadena, California, USA.

Krishnan, S. (2009). On the modeling of elastic and inelastic, critical- and post-buckling behavior of slender columns and bracing members. Technical Report EERL 2009-03, Earthquake Engineering Research Laboratory, California Institute of Technology, Pasadena, California, USA.

Krishnan, S. and J. F. Hall (2006a). Modeling steel frame buildings in three dimensions – Part I: Panel zone and plastic hinge beam elements. *Journal of Engineering Mechanics 132*(4), 345–358.

Krishnan, S. and J. F. Hall (2006b). Modeling steel frame buildings in three dimensions – Part II: Elastofiber beam element and examples. *Journal of Engineering Mechanics 132*(4), 359–374.

Mattiasson, K. (1980). Numerical results from large deflection beam and frame problems analyzed by means of elliptic integrals. *International Journal for Numerical Methods in Engineering 17*((1)), 145–153.

Oran, C. (1973a). Tangent stiffness in plane frames. *Journal of the Structural Division, ASCE 99*(6), 973–985.

Oran, C. (1973b). Tangent stiffness in space frames. *Journal of the Structural Division, ASCE 99*(6), 987–1001.

Pinkney, R. B. (1973). Cyclic plastic analysis of structural steel joints. Technical Report UCB/EERC-73-15, Earthquake Engineering Research Center, University of California, Berkeley, California.

Popov, E. P. and H. Petersson (1978). Cyclic metal plasticity: Experiments and theory. *Journal of the Engineering Mechanics Division, ASCE 104*(EM6), 1371–1388.